User Manual EdingCNC Software

Document Release 4.02

Published by: Bert Eding

Eindhoven
The Netherlands

Title: Eding CNC user Manual Software
Author: Bert Eding
Date: Friday, 04 December 2015

Document History

Version	Date	Author	Comment
1	2006-03-10	Bert Eding	Initial version
4.02.00	19-12-2014	Bert Eding	Updated for V4.02 Start with new history for V4.02
4.02.00	19-12-2014		3D printing finalized. Including support for Linear Delta systems.
4.02.00	19-12-2014		zWear and xWear (turning) added
4.02.00	19-12-2014		Chip load calculator added
4.02.00	19-12-2014		Inputs which generate ESTOP now latched
4.02.00	19-12-2014		Max long file mode value increased from 25000 to 100000
4.02.00	19-12-2014		iCNC600 CPU support added
4.02.00	19-12-2014		G0 acceleration and velocity factor added.
4.02.00	19-12-2014		User macro.cnc added, there are now 2 macro files.
4.02.00	19-12-2014		M48, M49, M50, M51, M52 feed and speed override control added
4.02.00	19-12-2014		User definable M-Functions and re-definable M functions added, you can me your own M3 with this.
4.02.00	19-12-2014		Exec interpreter function allows to start external programs from the interpreter.
4.02.00	19-12-2015		G64 P.. R.. Q.. F.. The new PATH CONTROL functions here allow very FAST and constant speed foam milling.
4.02.00	19-12-2015		Firmware changes are latched ESTOP signals and use MACHINE ON virtual output that starts the watchdog signal, system-ready for iCNC600 board.
4.02.01	09-01-2015		Added shortcuts ctrl+alt+s and ctrl+alt+f, see shortcut table.

4.02.05	19-02-2015		New shortcuts to navigate directly to the F-Key menu's, see keyboards shortcuts table.
4.02.12	28-03-2015		Added optional stop M1
4.02.14	30-03-2015		Added safety relay function.
4.02.19	16-05-2015		Added safety relay function.
4.02.20	20-05-2015		Added #5152 Tool output on/off
4.02.25	02-07-2015		Minor text corrections
4.02.26	08-07-2015		Added variables to contain G68 rotation and G51 scaling, see system parameters/variables table
4.02.27	13-07-2015		Tool Wear changed to Delta, so ZDelta and XDelta. This is conform Siemens CNC. Note that the signa of Delta is the opposite from Wear. ZDelta negative will mill deeper as ZDelta 0. So a negative number indicates the tool Length is shorter.
4.02.28	04-09-2015		Home tandem textual update. Screens that show green Home LEDs updated, home LEDs are only YELLOW or OFF. Removed Typo's
4.02.33	17-10-2015		Added section in chapter **Run behavior during simulation and render** that explains how to make macro.cnc code work correctly in simulation and render mode.
4.02.37	1-12-2015		Small text improvements

© Copyright Eding CNC Holding B.V.

All rights reserved. Reproduction in whole or in part prohibited without the prior written consent of the copyright owner.

ACKNOWLEDGEMENTS

The G-Code part of this user manual has been derived from the full report of the RS274/NGC language. Parts that are less relevant to EdingCNC users or parts that are not supported are left out.

Table of contents

Table of contents 5

1 Introduction 10

1.1 Context and scope 11

1.2 Definitions, acronyms and abbreviations 12

1.3 Minimum PC requirements 13

1.4 Installation of EDINGCNC 14

1.4.1 USB 14
1.4.2 Ethernet 15
1.4.3 Set admin mode 20
1.4.4 Profiles, If you have different setups 22

2 The user interface 24

2.1.1 Setup Page's 28
2.1.2 UI and Connection 28
2.1.3 Motor setup 29
2.1.4 Homing and ESTOP setup 31
2.1.5 Backlash setup 32
2.1.6 LAF setup 33
2.1.7 Kinematic Setup 36
2.1.8 Tool change Area setup 36
2.1.9 Tangential knife setup 36
2.1.10 Safety/Door open Input setup 39
2.1.11 Spindle and PWM setup 39
2.1.12 UI setup items 41
2.1.13 IO setup 42
2.1.14 Interpreter settings 42
2.1.15 Traffic light setup 43
2.1.16 JobTimeEstimation 44
2.1.17 Hand wheel Setup 44
2.1.18 Load/Run Automatically 45
2.1.19 Probing Setup 47
2.1.20 Camera Setup 48
2.1.21 CPUOPT 49
2.1.22 Using a safety relay 50
2.1.22.1 Powering the safety relay 50
2.1.22.2 Input contacts of the safety relay 50
2.1.22.3 Output contacts of the safety relais 51
2.1.22.4 Switching on the safety relay 51

2.2 Operate Page, this is where the machine is operated for milling 53

2.2.1 Operate page introduction 53
2.2.2 Reset Button F1 54
2.2.3 Escape Button 54
2.2.4 The menu's 55
2.2.4.1 Main Menu 55
2.2.4.2 Home menu 55
2.2.4.3 Zero menu 55
2.2.4.4 Auto menu 55

2.2.4.5 IO menu 61
2.2.4.6 Graphic menu 61
2.2.4.7 Jog menu 62
2.2.4.8 Jog pad 62
2.2.4.9 User menu 63
2.2.5 Operate page tasks 64
2.2.5.1 Startup 64
2.2.5.2 Homing 64
2.2.5.3 Load and run a g-code file 64
2.2.5.4 Mapping X/Y G-code to cylinder on A-Axis 69
2.2.5.5 Milling un-even surfaces 71
2.3 Linear PITCH compensation 77
2.4 Speed-PWM compensation 78
2.5 Program Page, DXF and HPGL import 80
2.6 Tools Page 83
2.6.1 Milling 83
2.6.2 Turning 84
2.6.3 Tool change 84
2.6.4 Automatic user defined Tool change ATC 84
2.6.5 The variable Page 85
2.7 IO Page 86
2.8 Service Page 87
2.9 Util Page, Chipload and Feed/Speed 88
2.10 homing and coordinate systems 89
2.10.1 Manual homing the machine 90
2.10.2 Automatic homing the machine and HomeIsEstop 91
2.10.3 Tandem axes homing 92
2.10.4 Work versus Machine coordinate system and zeroing 94
2.11 3D Printing 95
2.12 Keyboard shortcuts 98
2.13 Zero tool macro 100
2.14 Tool measurement Macro 101
3 Input: the RS274/NGC Language 105
3.1 System-Parameters/Variables 106
3.2 Tool data 110
3.2.1.1 Tool Orientation for lathes 110
3.3 Coordinate Systems 111
3.4 Format of a Line 112
3.4.1 Line Number 112
3.4.2 Word 112
3.4.2.1 Number 113
3.4.2.2 Parameter Value 113
3.4.2.3 Expressions and Binary Operations 114
3.4.2.4 Unary Operation Value 114
3.4.3 Parameter Setting 115
3.4.4 Comments and Messages 115

3.4.5 Item Repeats 115
3.4.6 Item order 116
3.4.7 Commands and Machine Modes 116

3.5 Modal Groups 117

3.6 G Codes 119
3.6.1 Rapid Linear Motion - G0 119
3.6.2 Linear Motion at Feed Rate - G1 121
3.6.3 Arc at Feed Rate - G2 and G3 122
3.6.3.1 Radius Format Arc 122
3.6.3.2 Center Format Arc 123
3.6.4 Dwell - G4 124
3.6.5 Set Coordinate System Data -G10 124
3.6.6 Plane Selection - G17, G18, and G19 124
3.6.7 Length Units - G20/G21 and G70/G71 124
3.6.8 Return to Home - G28 and G30 125
3.6.9 G33, G33.1 Spindle-Synchronized Motion 126
3.6.10 Straight Probe - G38.2 127
3.6.10.1 The Straight Probe Command 127
3.6.10.2 Using the Straight Probe Command 127
3.6.10.3 Example Code 128
3.6.11 Cutter Radius Compensation - G40, G41, G41.1, G42, G42.1 129
3.6.11.1 Example code for milling 130
3.6.11.2 Example code for turning 131
3.6.12 Tool Length Offsets - G43, G43 H, G43.1, and G49 133
3.6.13 Scaling G50/G51 133
3.6.14 Move in Absolute Coordinates - G53 134
3.6.15 Select Coordinate System - G54 to G59.3 134
3.6.16 Set Path Control Mode - G61, and G64 135
3.6.17 Coordinate system rotation G68 139
3.6.18 Threading (Lathe) – G76 139
3.6.19 Cancel Modal Motion - G80 142
3.6.20 Canned Cycles - G81 to G89 142
3.6.20.1 Preliminary and In-Between Motion 143
3.6.20.2 G81 Cycle 144
3.6.20.3 G82 Cycle 145
3.6.20.4 G83 Cycle 145
3.6.20.5 G73 Cycle 145
3.6.20.6 G84 Cycle 146
3.6.20.7 G74 Cycle 146
3.6.20.8 G85 Cycle 146
3.6.20.9 G86 Cycle 146
3.6.20.10 G87 Cycle 147
3.6.20.11 G88 Cycle 148
3.6.20.12 G89 Cycle 148
3.6.21 Set Distance Mode - G90 and G91 148
3.6.22 Coordinate System Offsets - G92, G92.1, G92.2, G92.3 149
3.6.23 Set Feed Rate Mode - G93, G94, G95 150
3.6.24 Spindle Control Mode – G96, G97 150
3.6.25 Set Canned Cycle Return Level - G98 and G99 150

3.7 Input M Codes 151
3.7.1 Program Stopping and Ending - M0, M1, M2, M30, M60 152
3.7.2 Spindle/Head Control - M3, M4, M5, M90-M97 152
3.7.3 Tool Change - M6 154
3.7.4 Coolant Control - M7, M8, M9 155
3.7.5 Feed-Speed Override Control - M48-M53 156

3.7.6 IO M Functions 156
3.7.7 Standard CNC IO - M3..M9, M80..M87 156
3.7.8 General purpose IO of CPU5B - M54, M55 and M56 156
3.7.9 A axis clamping M26, M27 158
3.7.10 Torch height control M20, M21 159
3.7.11 M Functions for 3D printing 159
3.7.12 M Function override and user m-functions 159

3.8 Other Input Codes 161
3.8.1 Set Feed Rate - F 161
3.8.2 Set Spindle Speed - S 161
3.8.3 Select Tool - T 161

3.9 Order of Execution 162

4 Language extensions 163

4.1 Flow control 164

4.2 supported operations on expressions 165
4.2.1 unary operations 165
4.2.2 binary operations: 165
4.2.3 An example: 166
4.2.4 Special interpreter commands, non G-Code 166
4.2.4.1 ErrMsg 166
4.2.4.2 Store position 166
4.2.4.3 DlgMsg 166
4.2.4.4 LogFile, LogMsg 169
4.2.4.5 TCAGuard [on | off] 169
4.2.4.6 MCAGuard [on | off] 169
4.2.4.7 HomeIsEstop [on | off] 169
4.2.4.8 Exec <external Program> <"parameter"> <Timeout In Ms.> 169

4.3 Macro file and automatic tool change 170
4.3.1 Tool change example 171
4.3.2 USER Reset 176

4.4 Run behavior during simulation and render 177
4.4.1 Example a check with error that we want to see always 177
4.4.2 Example a check with error showing only when running 177

5 Cutter Radius Compensation 178

5.1 Introduction 179
5.1.1 Data for Cutter Radius Compensation 180

5.2 Programming Instructions 181
5.2.1 Turning Cutter Radius Compensation On 181
5.2.2 Turning Cutter Radius Compensation Off 181
5.2.3 Sequencing 181
5.2.4 Use of D Number 181
5.2.5 Material Edge Contour 181
5.2.6 Programming Entry Moves 182
A.1.1.1 General Method 182
A.1.1.2 Simple Method 184

5.3 Nominal Path Contour 186

5.4 Programming Errors and Limitations 189

5.5 Extension IO board 192

5.6 Hardware installation tips..... 195

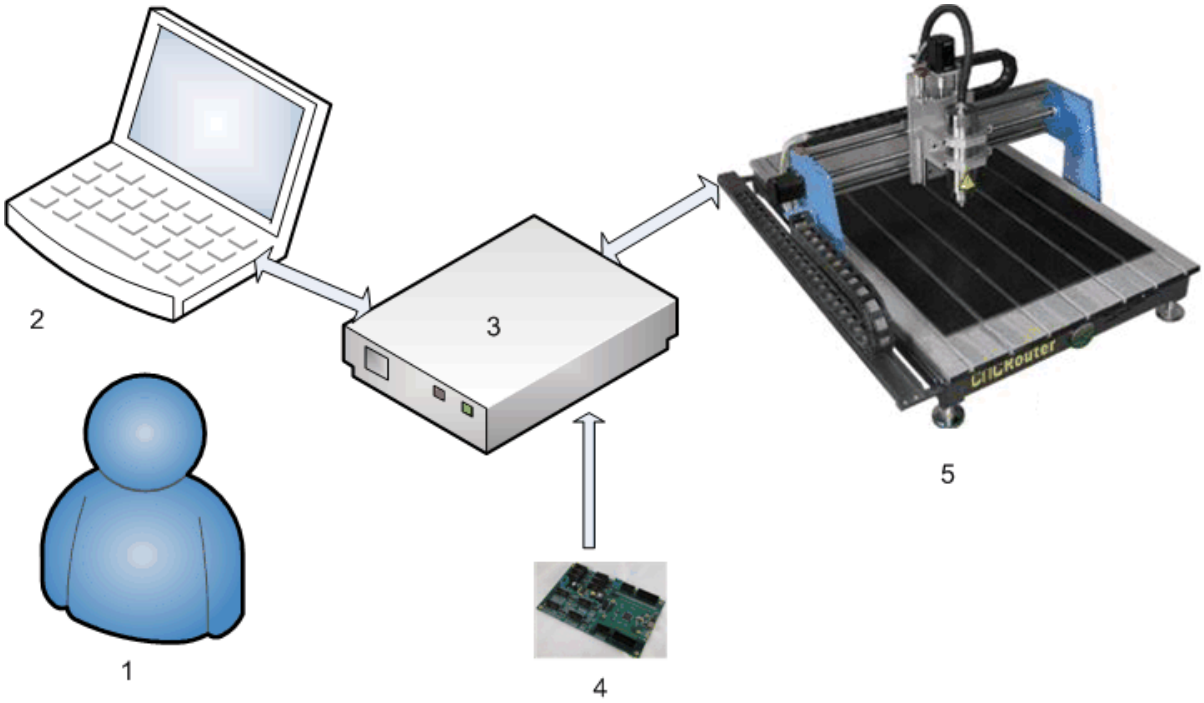
5.7 References..... 198

1 Introduction

This manual describes the usage of the CNC control system. Most hardware details can be found in the hardware documentation on the Eding CNC download page.

1.1 CONTEXT AND SCOPE

This section describes the context, hardware and software of a EDINGCNC controlled Machine.



1. Operator
2. PC connected via USB or Ethernet to electronic cabinet which contains the EDINGCNC CPU.
   The PC runs the EDINGCNC Control Software.
3. Electronics cabinet, with power supplies, drives and Eding CNC CPU inside.
4. EDINGCNC CPU
5. CNC Machine

The connection from CPU to the PC is USB or Ethernet depending on the CPU model.

The CPU delivers STEP/Direction signals to the power stage of each motor (drive), the motor connections of the drive go to the motors inside the machine.

Other connections like home-sensor/switches go directly from CPU to the machine.
For detailed info on all IO signals see the info in the technical flyers of the CPU, available on the download page.

The Scope of the EDINGCNC product is the EDINGCNC software on the PC and the EDINGCNC CPU.

1.2 DEFINITIONS, ACRONYMS AND ABBREVIATIONS

CNC	Computerized Numerical Control
CPU	Central Processor Unit, a PCB board with a Processor on it.
DXF	Drawing Exchange Format) is a CAD data file format developed by Autodesk
FIFO	First In First Out Buffer
HPGL	Hewlet Packard Graphical Language
GUI/UI	Graphical User Interface
INTERPRETER	A software function that is able to read a text file and execute the commands contained therein.
JOBFILE	A *job* is the text file (G code) that will be executed by the interpreter.
GUI	Graphical User Interface.
PWM	Pulse Width Modulation
G-Code	CNC specific language to control the movements and IO of a milling machine.
LAF	Look Ahead Feed, advanced motion algorithm that ensures minimal machining time.

1.3 MINIMUM PC REQUIREMENTS

- 1.4 GHz Atom.
- Pentium, duo-core recommended for Ethernet.
- 1024 MB RAM for XP, 4G for Windows 7/8.
- Windows XP or Windows 7/8, 32 or 64 bit.
- Minimum Screen resolution 1024 x 768.
- Graphic card with Open GL support is preferred.
- USB-2 connection / Ethernet connection for Ethernet CPU's
- Intel 100Mbit Ethernet card for Ethernet CPU's.

Windows XP and Windows 7 and Windows * is proven to work fine with EdingCNC.
Windows Vista is not.

1.4 INSTALLATION OF EDINGCNC

Download the installation executable from the website download page. Click on it to install the software. Follow the screens. On Windows 7, click with the right mouse button, start as administrator.

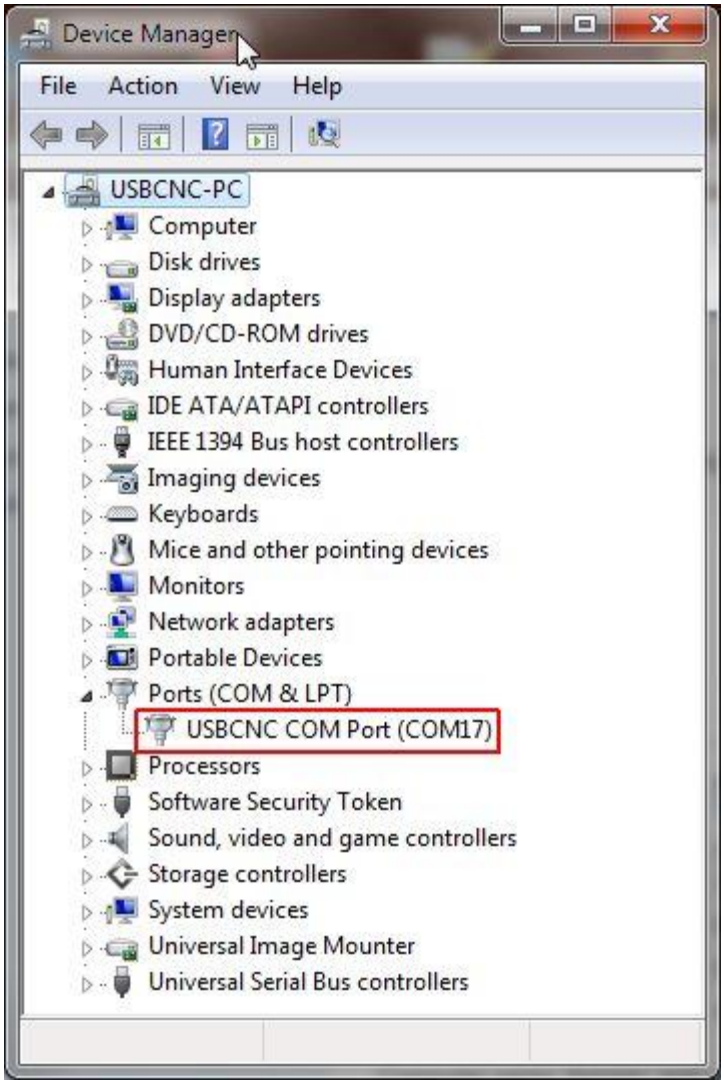
For setup of the hardware, check the hardware technical flyers for your CPU type. They are on the download page of the website.

1.4.1 USB

During installation be sure to check "Install USB drivers":

Setup - USBCNC4Beta
Completing the USBCNC4Beta Setup Wizard
Setup has finished installing USBCNC4Beta on your computer. The application may be launched by selecting the installed icons.
Click Finish to exit Setup.
Install USB drivers
View the ReleaseNotes.txt file
Finish

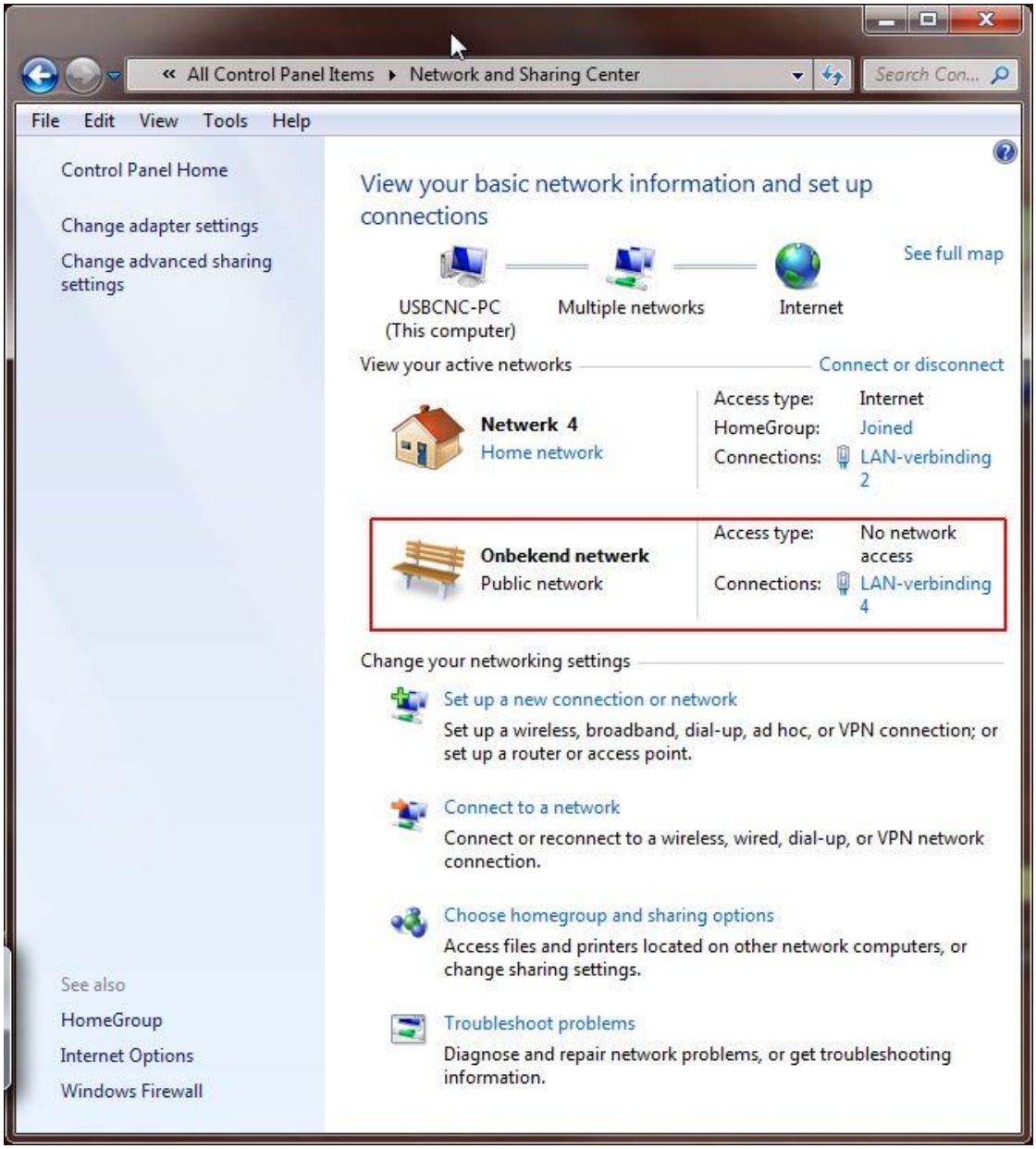
After installation reboot the PC, when it is rebooted connect the CPU, after 10 – 60 seconds, you will see that windows has found an EDINGCNC COM port if you are using and USB based CPU board. You can check that the USB driver is correctly installed in windows device manager, press Windows start button==>my computer, click with right mouse button and select "properties".
Select Device Manager:



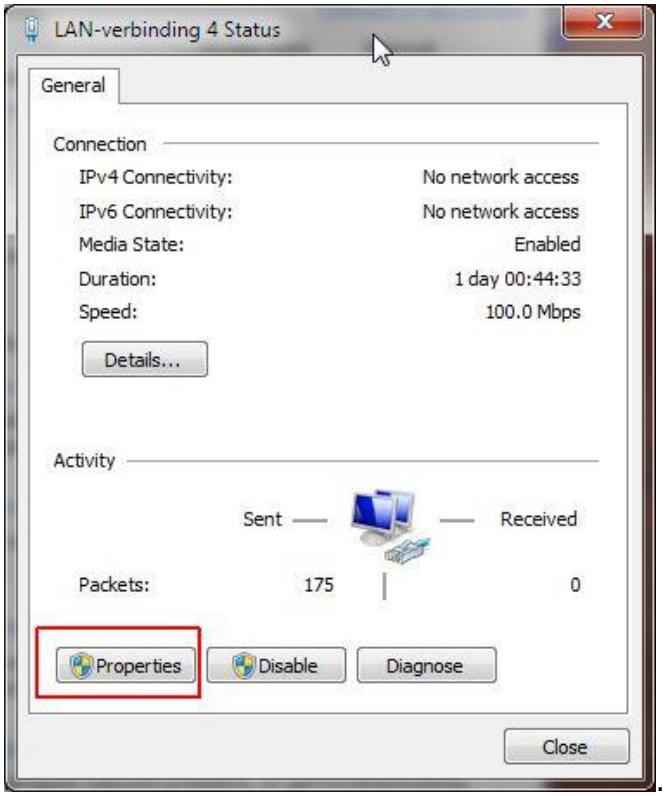
If you see this, the USB driver is correctly installed. The COM17 number may be different on your system.

1.4.2 **Ethernet**

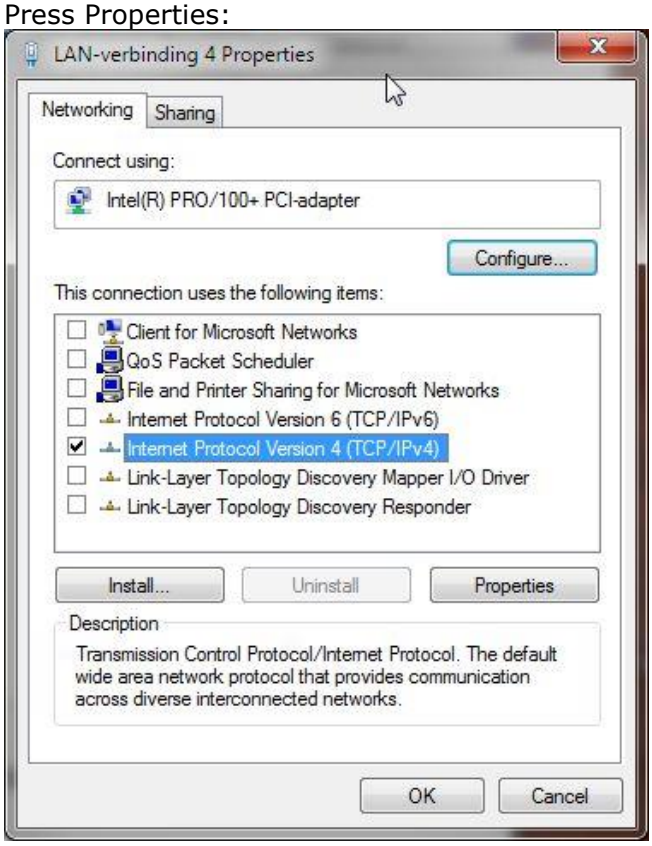
For Ethernet you need a free Ethernet connection on the PC, Add a 2nd network card if needed. Connect the CPU using a 100 MBit UTP Cross cable. Then setup the Ethernet adapter. Go to the windows network settings, the network adapter with No network access is one for the CPU:


« All Control Panel Items ▸ Network and Sharing Center
Search Con...
File Edit View Tools Help
Control Panel Home
Change adapter settings
Change advanced sharing settings
View your basic network information and set up connections
See full map
USBCNC-PC (This computer)
Multiple networks
Internet
View your active networks
Connect or disconnect
Netwerk 4
Home network
Access type: Internet
HomeGroup: Joined
Connections: LAN-verbinding 2
Onbekend netwerk
Public network
Access type: No network access
Connections: LAN-verbinding 4
Change your networking settings
Set up a new connection or network
Set up a wireless, broadband, dial-up, ad hoc, or VPN connection; or set up a router or access point.
Connect to a network
Connect or reconnect to a wireless, wired, dial-up, or VPN network connection.
Choose homegroup and sharing options
Access files and printers located on other network computers, or change sharing settings.
Troubleshoot problems
Diagnose and repair network problems, or get troubleshooting information.
See also
HomeGroup
Internet Options
Windows Firewall

Click on the adapter with no network access, here: "LAN-verbinding 4" here, the text in your PC may be different.

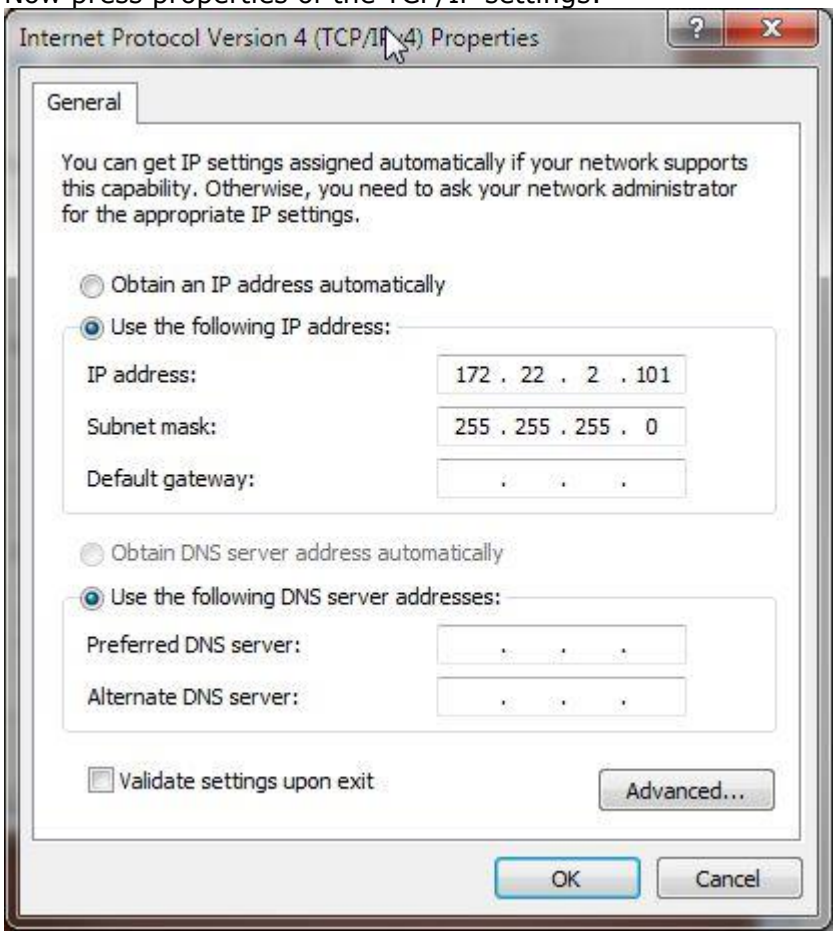


Press Properties:



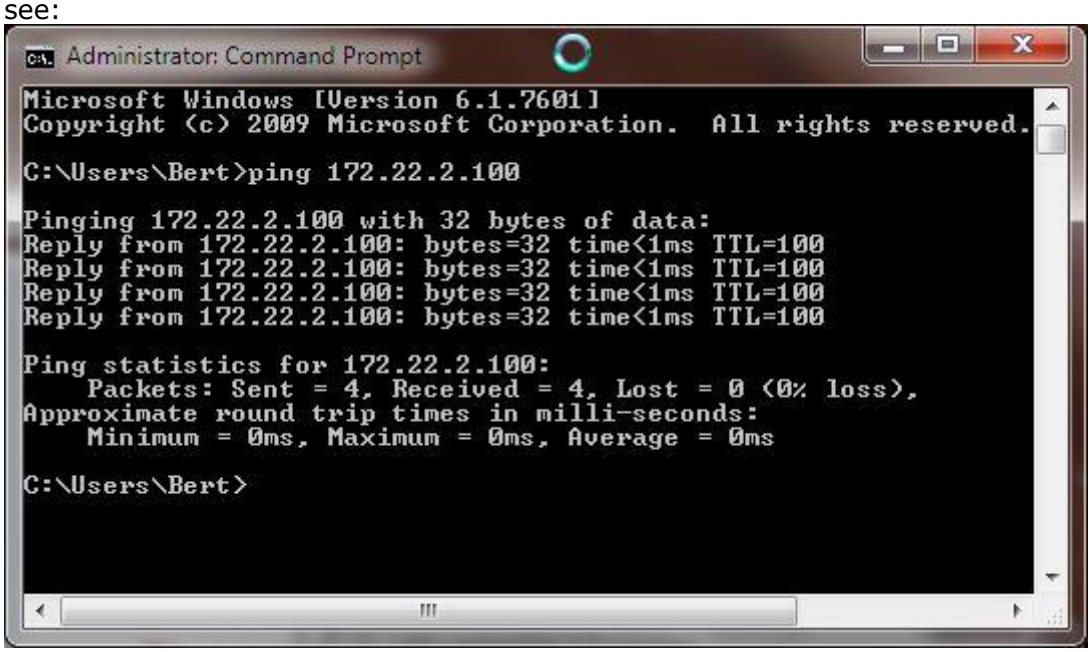
Switch on only TCP/IP V4 and uncheck the rest.

Now press properties of the TCP/IP settings:

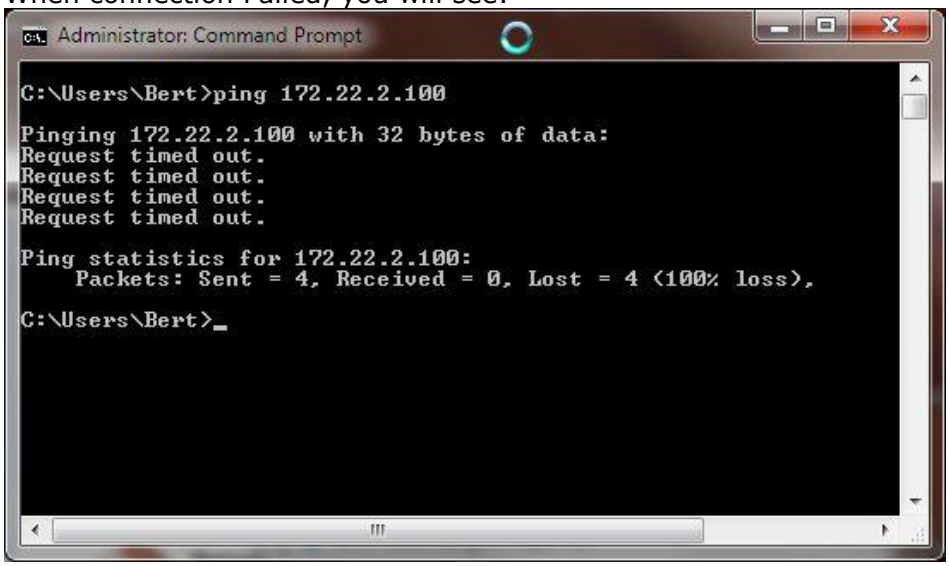

Internet Protocol Version 4 (TCP/IPv4) Properties
General
You can get IP settings assigned automatically if your network supports this capability. Otherwise, you need to ask your network administrator for the appropriate IP settings.
Obtain an IP address automatically
Use the following IP address:
IP address: 172 . 22 . 2 . 101
Subnet mask: 255 . 255 . 255 . 0
Default gateway:
Obtain DNS server address automatically
Use the following DNS server addresses:
Preferred DNS server:
Alternate DNS server:
Validate settings upon exit
Advanced...
OK
Cancel

The PC LAN adapter gets IP Address 172.22.2.101.
The EDINGCNC CPU network is setup for 172.22.2.100.

Press OK, now you can test if the network is working, click the Windows Start button, select all programs ==> accessories==>command prompt.
In the command prompt, enter ping 172.22.2.100, when connection OK, you should see:


Administrator: Command Prompt
Microsoft Windows [Version 6.1.7601]
Copyright (c) 2009 Microsoft Corporation. All rights reserved.
C:\Users\Bert>ping 172.22.2.100
Pinging 172.22.2.100 with 32 bytes of data:
Reply from 172.22.2.100: bytes=32 time<1ms TTL=100
Reply from 172.22.2.100: bytes=32 time<1ms TTL=100
Reply from 172.22.2.100: bytes=32 time<1ms TTL=100
Reply from 172.22.2.100: bytes=32 time<1ms TTL=100
Ping statistics for 172.22.2.100:
Packets: Sent = 4, Received = 4, Lost = 0 (0% loss),
Approximate round trip times in milli-seconds:
Minimum = 0ms, Maximum = 0ms, Average = 0ms
C:\Users\Bert>

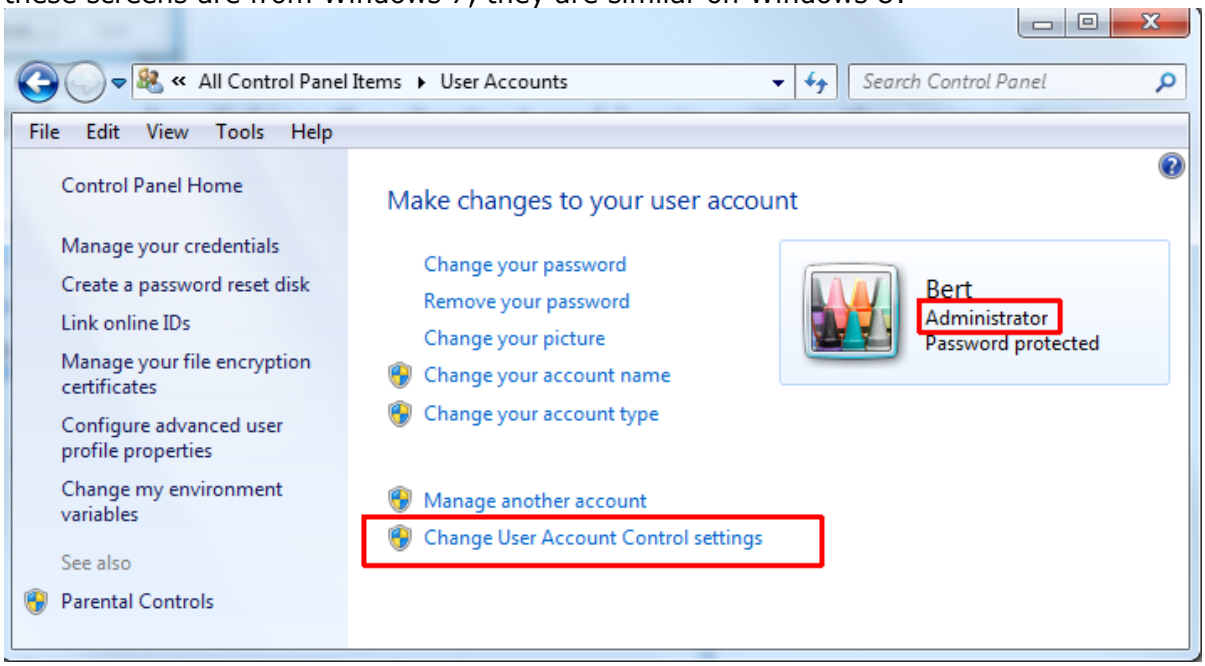
When connection Failed, you will see:


Administrator: Command Prompt
C:\Users\Bert>ping 172.22.2.100
Pinging 172.22.2.100 with 32 bytes of data:
Request timed out.
Request timed out.
Request timed out.
Request timed out.
Ping statistics for 172.22.2.100:
Packets: Sent = 4, Received = 0, Lost = 4 (100% loss),
C:\Users\Bert>

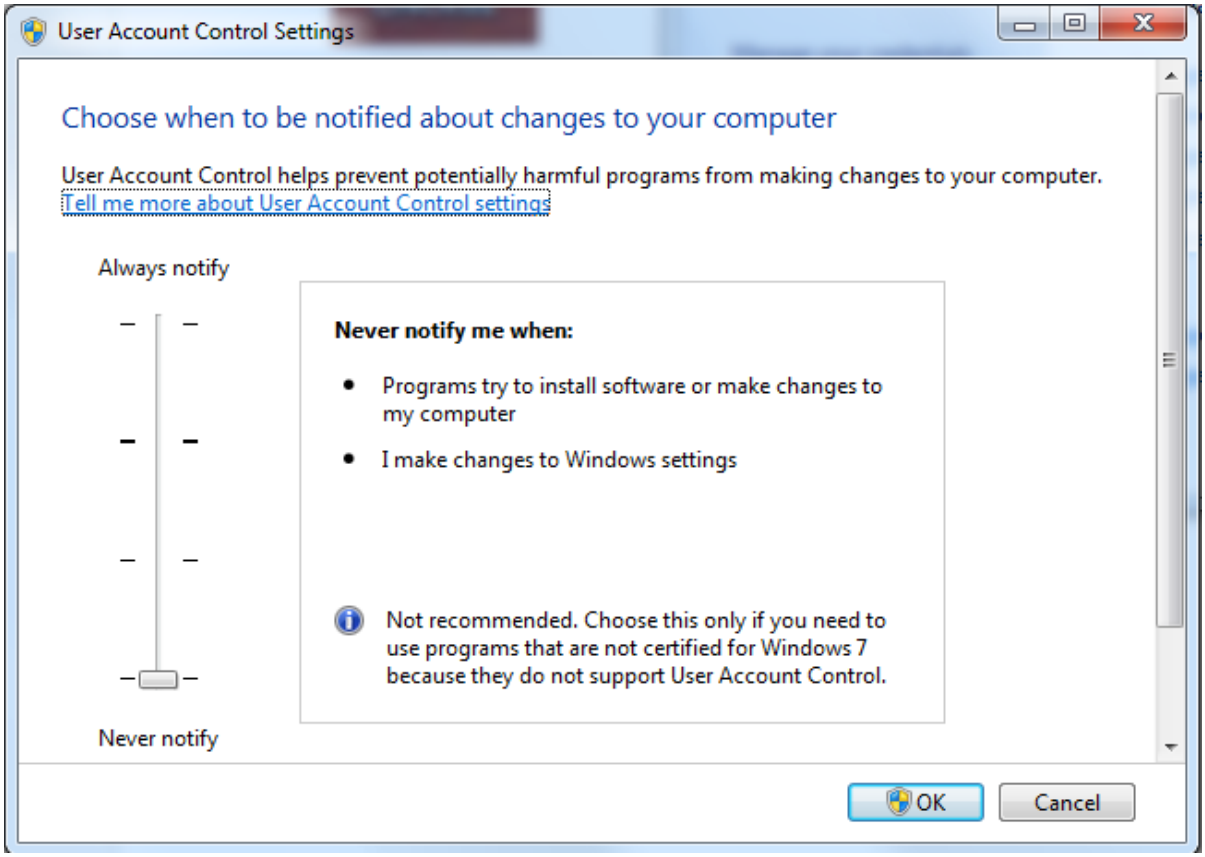
If you have this, check you cable and your network settings again. Also check that the yellow led on the CPU is flashing at approximately 1Hz.

1.4.3 **Set admin mode**

Eding CNC software needs real-time priority on Windows to control, your machine correctly. This only allowed with Admin rights, so make sure, the user has Admin Rights, these screens are from Windows 7, they are similar on Windows 8:

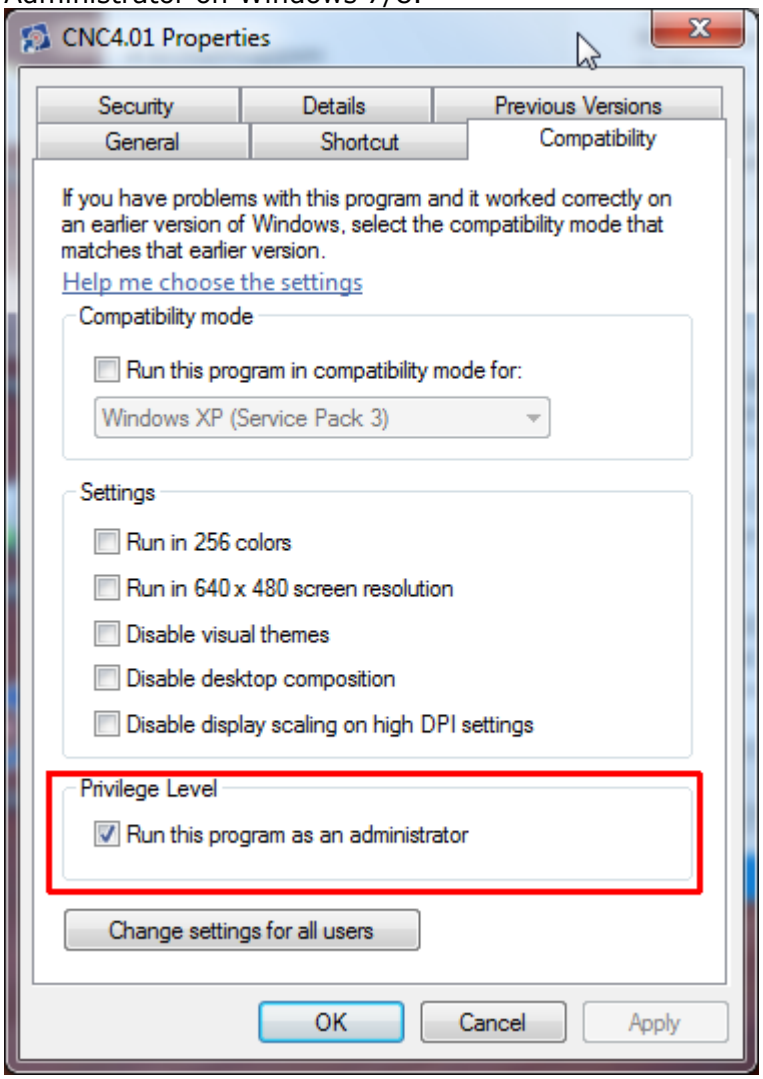


And switch off UAC.


User Account Control Settings
Choose when to be notified about changes to your computer
User Account Control helps prevent potentially harmful programs from making changes to your computer.
Tell me more about User Account Control settings
Always notify
Never notify
Never notify me when:
Programs try to install software or make changes to my computer
I make changes to Windows settings
Not recommended. Choose this only if you need to use programs that are not certified for Windows 7 because they do not support User Account Control.
OK
Cancel

After the install you will have the program ICON on your desktop. Do not start it yet, read further.

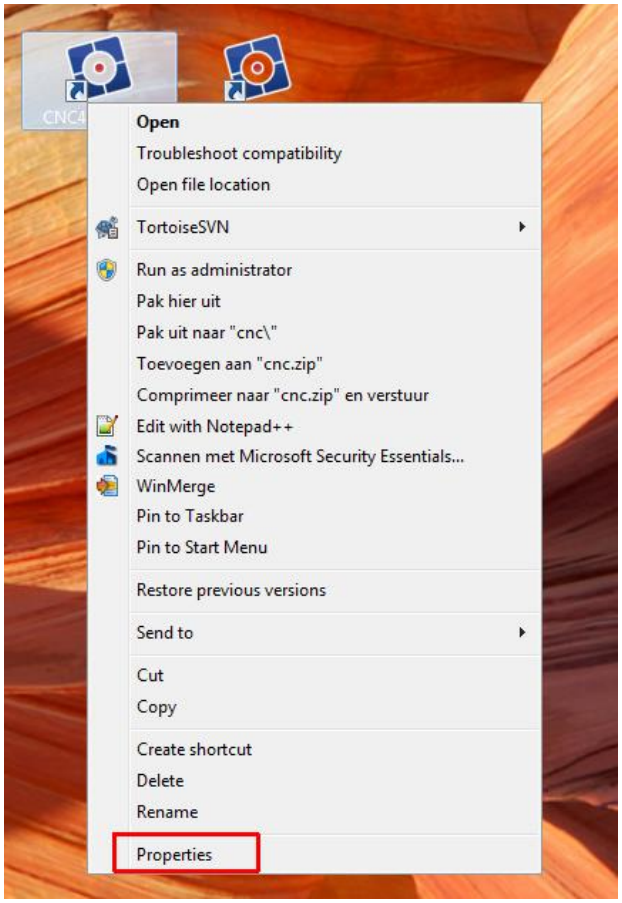
The application must be started as Administrator and the user starting it must have admin rights. Right click the mouse on the EdingCNC Icon and then select run as Administrator on Windows 7/8.


CNC4.01 Properties
Security
Details
Previous Versions
General
Shortcut
Compatibility
If you have problems with this program and it worked correctly on an earlier version of Windows, select the compatibility mode that matches that earlier version.
Help me choose the settings
Compatibility mode
Run this program in compatibility mode for:
Windows XP (Service Pack 3)
Settings
Run in 256 colors
Run in 640 x 480 screen resolution
Disable visual themes
Disable desktop composition
Disable display scaling on high DPI settings
Privilege Level
Run this program as an administrator
Change settings for all users
OK
Cancel
Apply

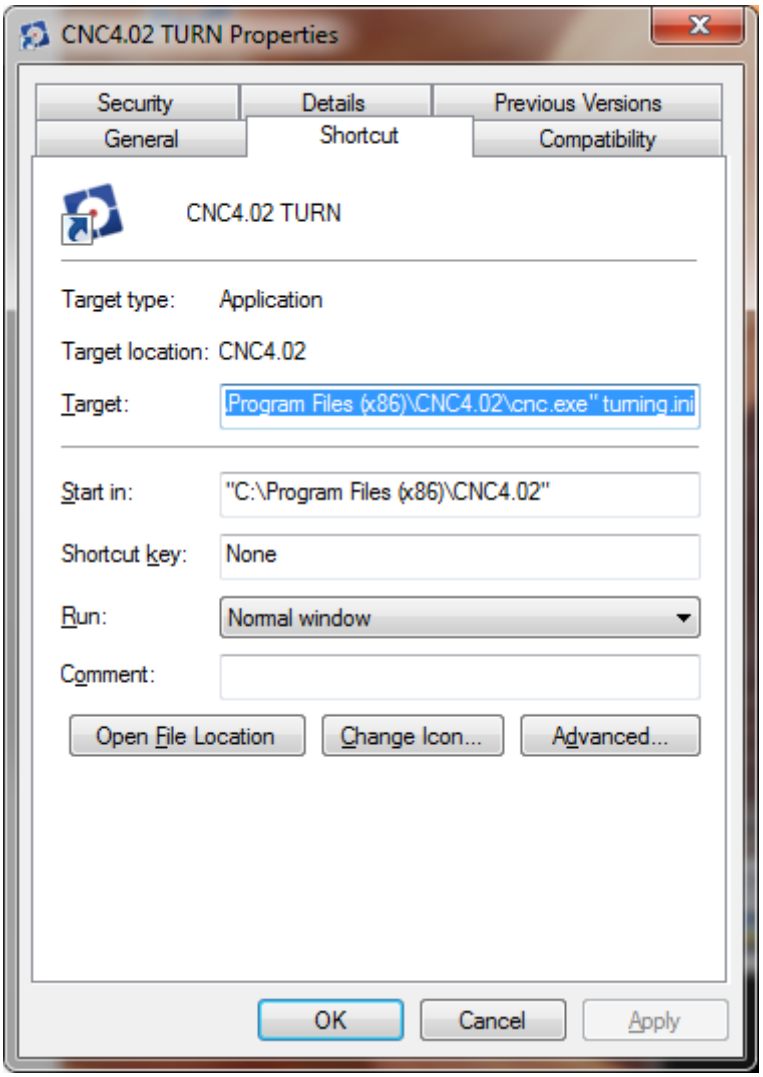
You can now start the software.

1.4.4 Profiles, If you have different setups

If you have e.g. a milling machine and a turning machine controlled from the same computer, you can make a copy of the software ICON and then rename one to “CNC4.02 TURN” and the other “CNC4.02 TURN”.
Now right click de ICON and select “properties”:



Then you must tell the software which settings file to use. This can be done by editing this line in the TAB "Shortcut"



Change this line "C:\Program Files (x86)\CNC4.02\cnc.exe", add turning.ini at the end:
"C:\Program Files (x86)\CNC4.02\cnc.exe" turning.ini

Now the software will use turning.ini to store your settings.
Without the turning.ini the software uses default file name cnc.ini to store the parameters.

2 The user interface

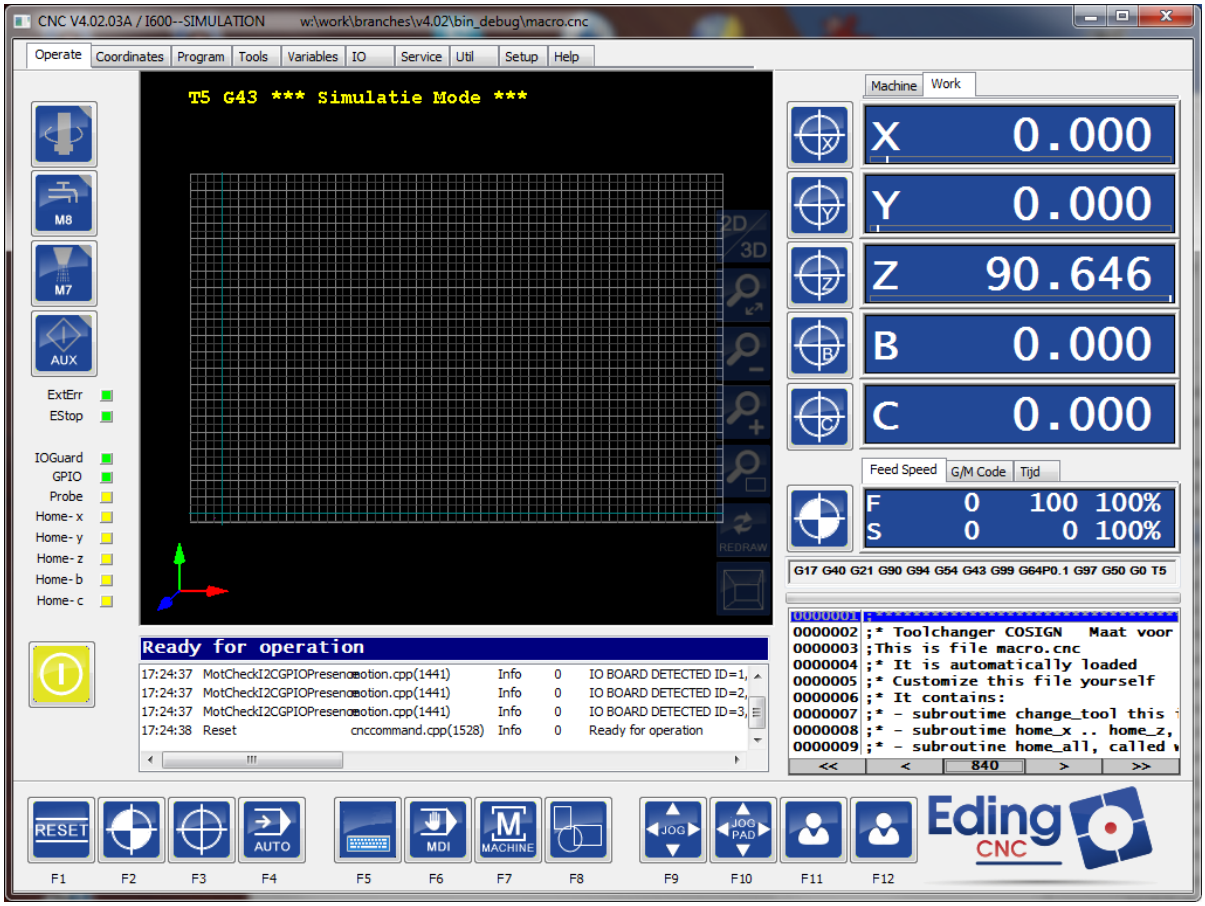
General info:

There are several views:
Operate , Program, Tools, Variables, Setup and Help.
Using control-tab, you can tab through them.

It is important that EDINGCNC is started started as administrator. On windows 7 this is not automatically done like in XP.
Click on the right mouse button and select "Run As administrator". You can also set this in the ICON properties, compatibility.

When you start EDINGCNC for the first time you will get the Terms/Guarantee page click the language you read the text. Then click agree if you agree. The operate page is shown. This is the main screen to do all machine operation's from.
Running a program and fast jogging is only possible after the machine is correctly homed, so this must be setup first. The reason is that collision prevention is not active when the machine isn't homed, so damage to the machine may happen when homing is not performed.

Program window "Operate" is shown All machine operation are performed from this TAB.

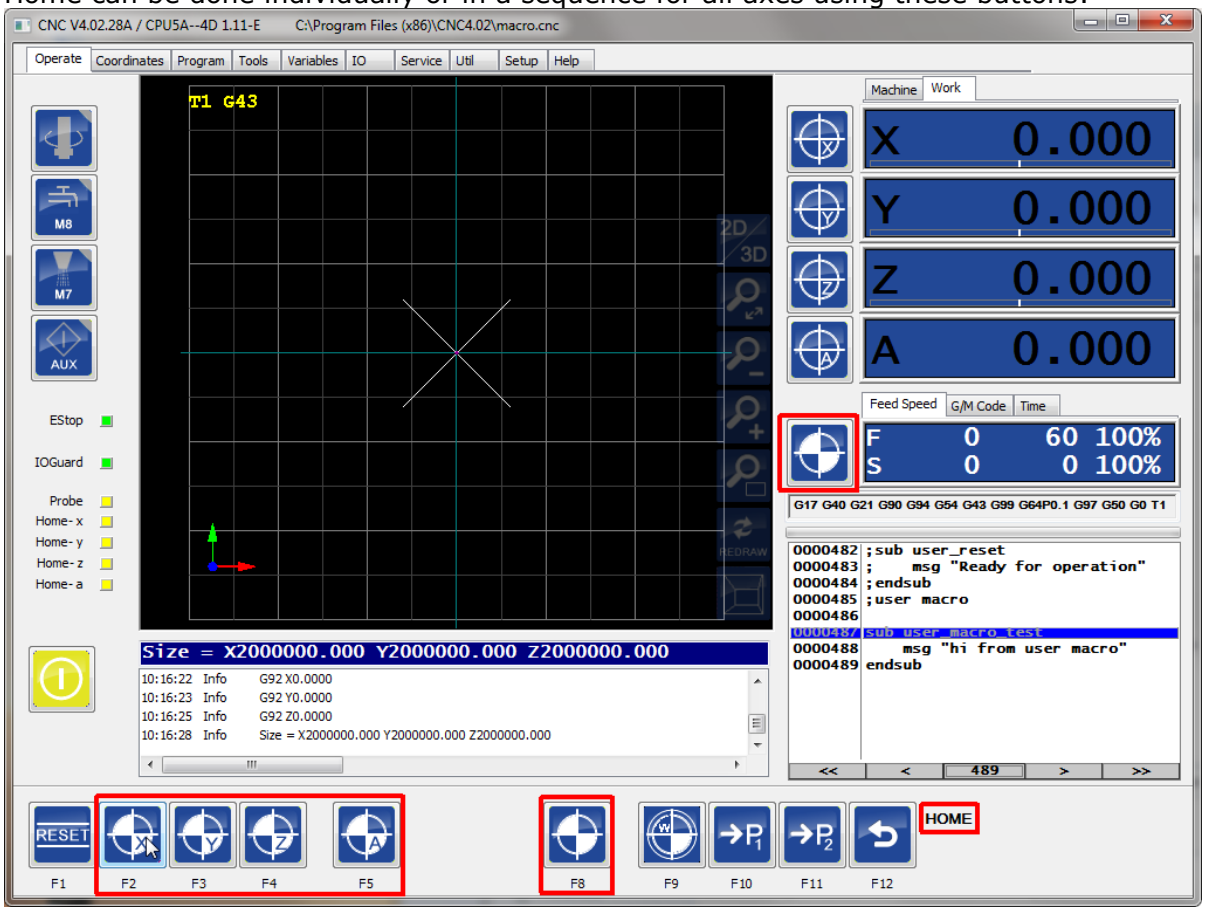


If all settings for your machine you can press Reset (F1), this enables your drives.
Then you are asked to home the axes:

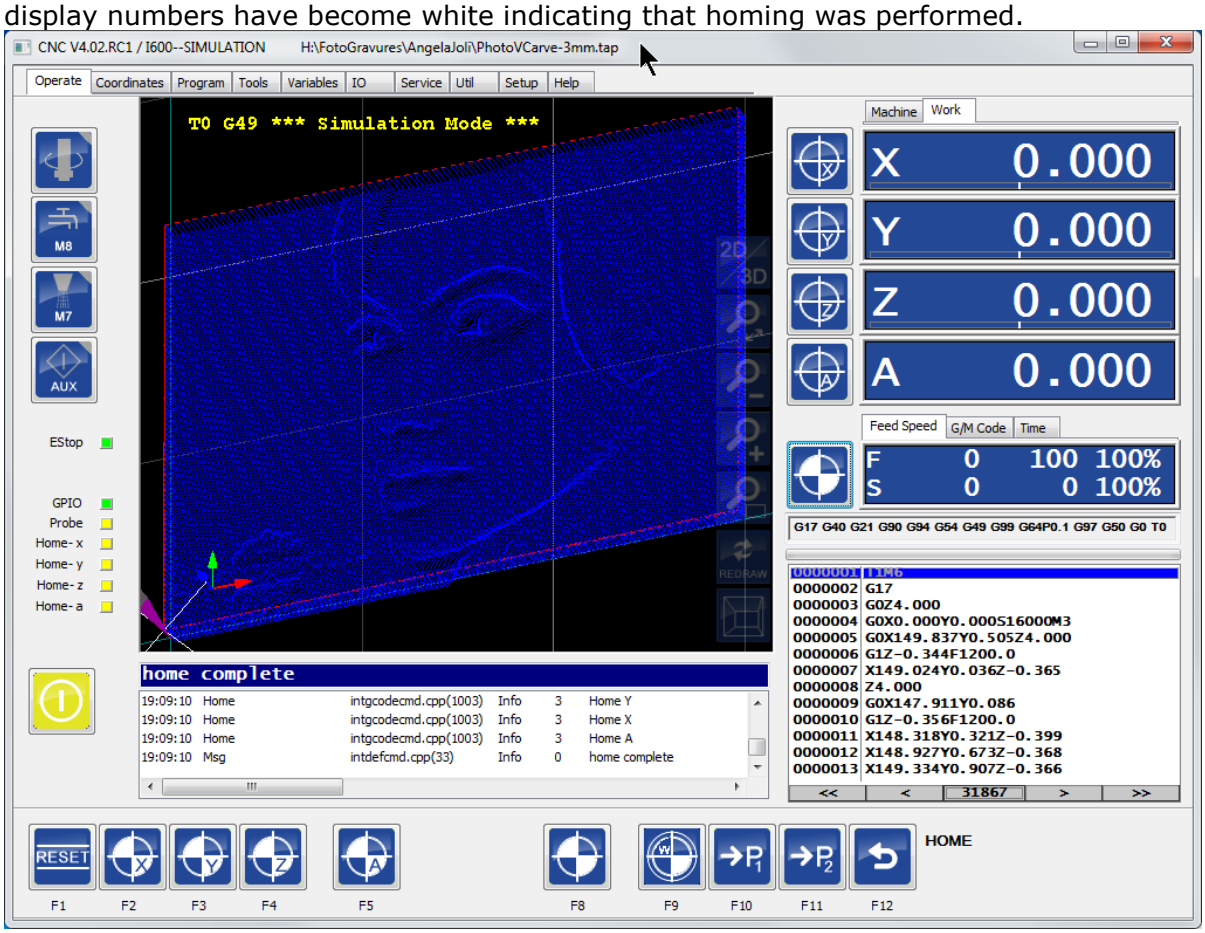
When you see this, you can use the home menu or the button besides the Feed Speed readout. Buttons you need often like this one is always nearby.

Homing is very important, after homing the position inside the software does match with the physical machine. So only then the software will allow movements correctly and stop just before the limits of the machine without collision against the machine limits.
So always make sure homing is working correctly!

Home can be done individually or in a sequence for all axes using these buttons:



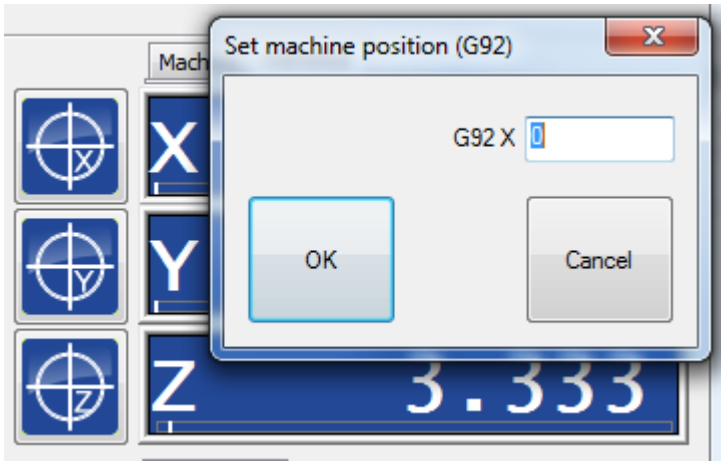
When homing is done you are free to move the axes by the JOG Keys, the position display numbers have become white indicating that homing was performed.



These are the arrow keys on your keyboard:

With the keys you move the axes, we call this jogging.
Using the keys alone you have low speed, 10%.
Using the keys in combination with the CTRL key, you get 50% speed.
You get 100% speed in combination with the SHIFT key.

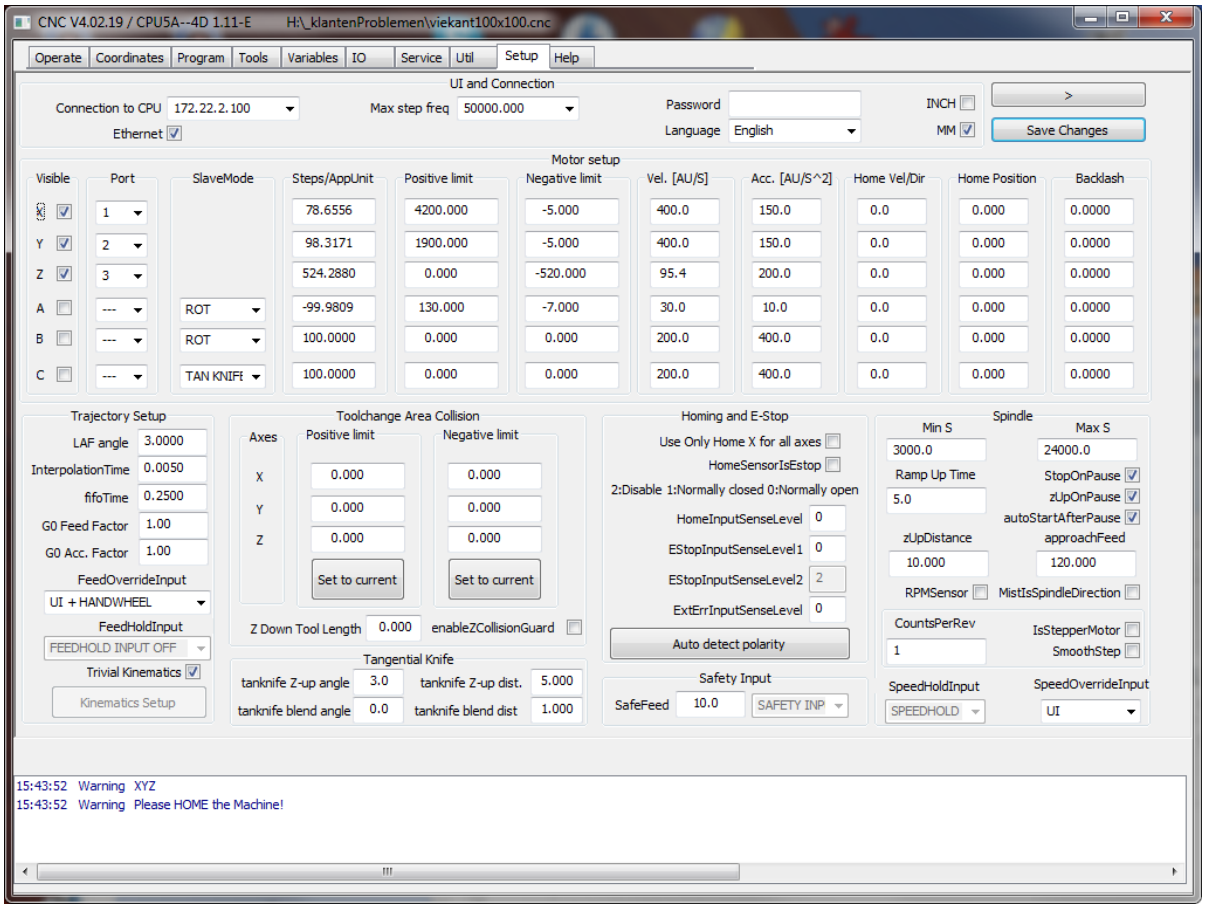
You can set the axes position by clicking the axis position display:



And you can also move the axes by holding the CTRL key and clicking with the left mouse button on the position display.

2.1.1 **Setup Page's**

Before the system is actually used we have to setup the system to accommodate the machine, we do that in the setup page's. There a two main setup pages:



2.1.2 **UI and Connection**

Connection to CPU: If you have 1 board connected to your PC, leave the setting at AUTO, the software will find the board automatically. Otherwise choose the here the CPU you want to work with. For CPU's with USB, you see the COMx ports here, in case of a CPU5 with Ethernet, you will see the IP-Address here.

Ethernet: If you have a CPU with Ethernet, check the Ethernet checkbox.

Max Step Frequency:

The maximum step frequency that the CPU will generate. For motor drives it is needed to lower the maximum frequency. In case the drive is unable to handle the high step frequency or low step-pulse width.

Some of the digital drives from Leadshine cannot handle 125Khz step-rate, they need a setting of 90Khz or lower.

Language setup: Speaks for itself. After it is set, save the changes, then close EDINGCNC and restart so that everything will be in the correct language. The translations are in 2 files, cncgui-lang.txt and cncserver-lang.txt, if you find translation mistakes you can correct this here. Please send the corrected file to Eding CNC, the corrections will then be incorporated into new versions.

Password: You can protect the setup parameters from being modified by unauthorized persons by using a password. Leave empty if no password is desired.

INCH: Machine setup is in inch mode.
MM: Machine setup is in mm mode.

2.1.3 Motor setup

Visible: Check if the axis should be visible in the GUI.

Port: Map axis to a physical output port of the CPU.
If an axis is mapped and not visible, it can still be used in the interpreter.

Mode : Select mode for rotation axes, slave or special function:

- **ROT**, default, axis behaves as a normal rotation axis.
- **SLAVE** X, SLAVE Y or SLAVE Z axis is slave of X or Y or Z axes, for Gantry machines with two independent (Tandem) motors on the main axes. See also the Homing chapter for details on Slave axes.
- **FOAM CUT** for A-Axis, if used as a Foam cutter with 4 linear axes. X is the left horizontal axis, Y is the left vertical axis, A is the right horizontal axis and Z is the right vertical axis. Feed calculation are based on the X/Y or A/Z combination which ever makes the biggest distance,
- **4th MILL**, if used in 4 axes milling. Feed calculations are optimized such that the tooltip gets the correct speed relative to the material.
- **Tangential Knife**, this option is available for the C-Axis only. The Knife will rotate in the movement direction of X-Y. See also trajectory setup.
- **2nd Z**, for machines with 2 Z axes, where the A axis is used as the 2nd Z.

Steps/AppUnit :
Fill in number of steps per millimeter for millimeter mode or number of steps per inch for inch mode. For rotary axes, the unit is always steps per degree. Fill in a **negative number to reverse** the motor direction.
Example: Suppose your driver is set to 1600 steps/revolution (1/8 micro step) and you have coupled the motor directly to a spindle with 5mm pitch. The number to be filled in here = 1600 / 5 = 320.
If the movement direction is wrong, change it to -320.

Positive limit: Maximum machine position.
Negative limit: Minimum machine position;

Vel : Maximum axis velocity, all velocities, whether jogging, G0/G1/G2/G3 are limited to this value.

Acc: Maximum acceleration.
When this value is set equal to the Vel parameter, it will take 1 second to reach the max velocity. When the value is 2x the Vel parameter, the max velocity is reached in 0.5 second.

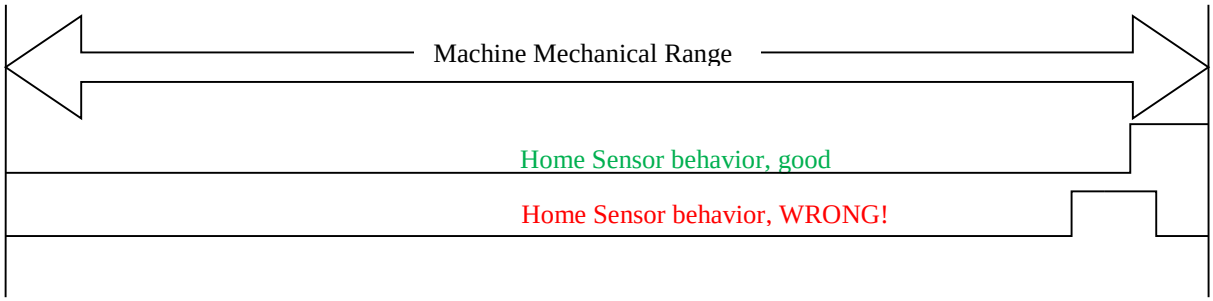
2.1.4 Homing and ESTOP setup

Home Vel/Dir: Homing velocity, a negative number reverses the homing **direction**.

When the velocity is set to 0, the axis is homed **manually**, see the homing and coordinate systems chapter.

Home Position: Machine position at the moment the home switch activated. This determines the machine coordinates. It is not really relevant where the machine zero point lies, it should only match with the MIN/MAX position.

Homing sensors should be setup such that they remain active until the mechanical end of the machine. The space from home sensor activation to mechanical end is required to ramp down the movement.



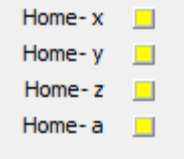
Use only home X for all axes:
Check this option if you have all home sensors wired to one input.

HomeSensorIsEStop:
The home sensors can also be used as limit switch which generate an E-Stop when activated. When this function is required, the sensors should be mounted outside the normal machine area.
Check this option if the home sensors work as EStop when activated. This option will work after homing is complete. The reason is that otherwise homing itself will generate an E-Stop.

Note that if this option is used and a home sensor triggers, it will be latched, so that it will remain visible in the UI also if it was triggered by a short pulse. The Reset will clear the latch.

HomeInputSenseLevel:
Defines HomeSensor input behavior,
0 = low active (normally open switch),
1 = high active, (normally closed switch).

set the level of your end of stroke switches, these are used for homing the machine. First check that the home sensors or switches are working, activate them and look at the home-LED's at the lower left side of the main Operate screen. If you see it working, take care that the machine axes are at the working area, so that none of the sensors are activated. Look at GUI "LEDs"



EStopInputSenseLevel1:
Defines EStop input behavior,
0 = low active (NO switch CPU5 series, NC switch iCNC600),
1 = high active, (NC switch CPU5 series, NO switch iCNC600).
2 = OFF, not used

EStopInputSenseLevel2, CPU5B only:
Defines EStop input behavior for second EStop input,
0 = low active (normally open switch),
1 = high active, (normally closed switch).
2 = OFF, not used

ExtErrInputSenseLevel, CPU5B and iCNC600 ONLY:
Defines External Error input behavior (CPU5B only),
0 = low active, e-stop (NO switch CPU5B, NC switch iCNC600),
1 = high active, e-stop (NC switch CPU5B, NC switch iCNC600).
2 = OFF, not used
3 = low active, smooth stop
4 = high active, smooth stop
With smooth stop the axes speed is ramped down, this means that there is no position loss.

The polarity settings for the home inputs, E-stop's and Extern error can be automatically determined by pressing the "Auto detect polarity" button.

Note that if the estop or extern-error input triggers, it will be latched, so that it will remain visible in the UI also if it was triggered by a short pulse. The Reset will clear the latch.

ExtErrInputSenseLevel via Sync input, CPU5A ONLY:
CPU5A do not have an ExtErr input, but applications that do need this function, can use the **Sync input** with limited functionality
Defines External Error input behavior (CPU5A only),
3 = low active, smooth stop
4 = high active, smooth stop
With smooth stop the axes speed is ramped down, this means that there is no position loss.

2.1.5 Backlash setup

Backlash: Set the amount of backlash for each axis that the software should compensate. Experiment with velocities and acceleration, the backlash compensation demands more from acceleration your motors than without backlash compensation. Do not try to compensate more than 0.1 millimeters. If there is more backlash, try to reduce it mechanically first.

2.1.6 **LAF setup**

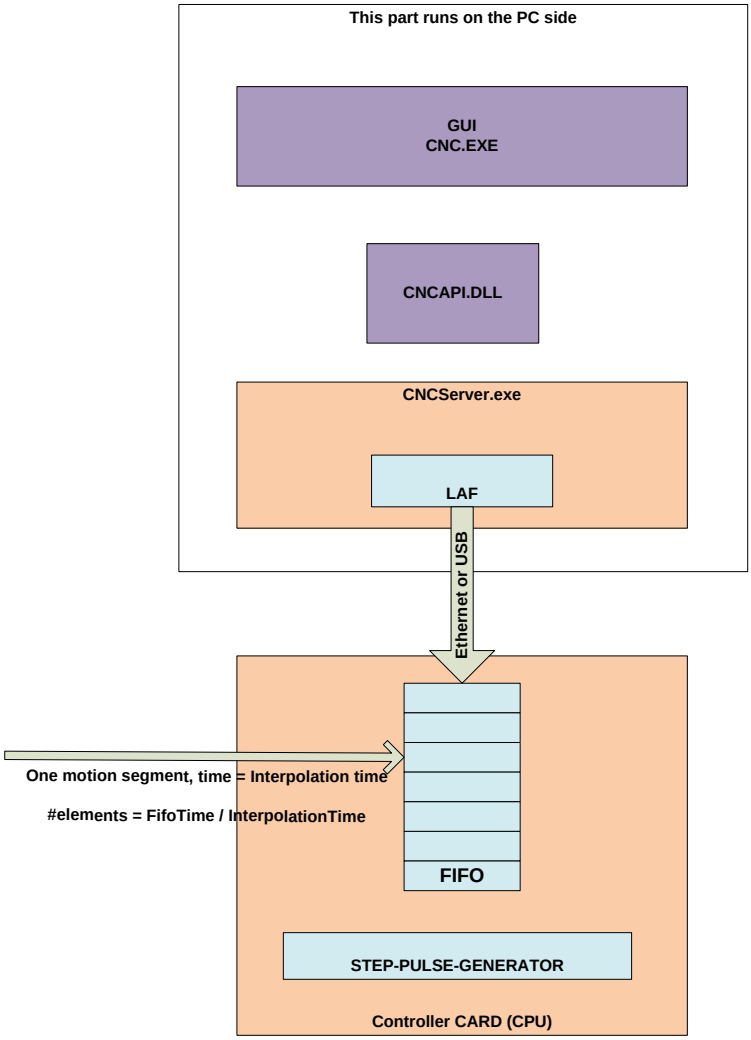
LAF minimum angle:

Look Ahead Feed calculations: Motion segments (g1,g2,g3) that are connected with a smaller angle as specified in **min.angle** will accelerate through which will give higher speeds especially with programs consisting of small motion segments. This is a unique feature which you don't find easily on low cost CNC controllers.
Be carefully with the min.angle setting because this cause acceleration spikes, it depends on your machine and the speed up till what extend this is possible. I suggest performing tests with en check whether you get step pulse loss.
A value of 0.1 .. 3 degrees is generally safe. Segments that are really tangential connected will move fast that way.

An example of what I use:
When using CorelDraw, a circle is drawn of 100mm in diameter, and exported as HPGL CorelDraw generates small line segments of approximately 6 degrees. Now I have set the min.angle to 6, this gives the possibility to mill the circle with a speed of F6000 while without LAF the speed would be approx F1300 on my machine.

InterpolationTime and FifoTime:

Every motion command is chopped up in small motion segments with a time of **InterpolationTime** in the setup.
The segments are send to the controller (CPU) which has a buffer (**FIFO**) that holds the motion segments. The step-pulse generator takes the motion segments one by one and generates stepper motor pulses from that. The number of the segments in the FIFO is depending on the **FifoTime** specified in the setup.



The **FIFO** makes it possible to perform smooth motion without hiccups on a non-real-time operating system like Windows. Because what happens is that sometimes Windows does things for itself stopping the execution of CNCSERVER.EXE for short times. This is no problem as long as the FIFO does not run empty (If the windows hiccup is longer than the FifoTime).

The default value for **InterpolationTime** = 0.005 seconds. The default FifoTime = 0.25 seconds. This means that the FIFO can hold 50 motion segments.

Why are these parameters adjustable?

1. For some applications it is desirable to have a lower interpolation time, e.g. if you have a machine with very high acceleration. a lower **InterpolationTime** may give smoother acceleration. The minimum interpolation time is 0.0025 second.
2. For some applications it is desirable to have lower **FifoTime**, e.g. Plasma THC, it will give more dynamic to the THC control if the **FifoTime** is lower.

Setting the **FifoTime** or **InterpolationTime** lower also means that the communication needs to be faster, so if these settings are modified to lower values, **a faster PC is required**.

FIFO UNDERRUN ERROR

If this happens then this indicates that the PC is too slow to keep the FIFO full.

Possible causes are:

- PC too slow.
    - The processor is too slow. (recommended minimum is 1.3 GHz duo-core processor, 2G RAM for 32bit Windows, 4G RAM for 64 bit Windows. If you execute large 3D g-code files > 10 Million lines, 4G RAM or more is recommended.
    - Too little memory in the PC (can be checked in task manager, there must always be Physical memory available, if the system starts swapping to disk because the memory is all used, changes on FIFO UNDERRUN is high).
    - PC has switched to energy saving mode and has become slow. So always adjust the energy saving settings such that there is maximum performance always.
    - Because you suddenly get an automatic Windows Update, so always switch off automatic Update and perform updates manually when you want.
    - Because virus checkers become active making the system slow. So always turn off anti-virus check when running the CNC controller.

- USB communication too slow or EMI disturbance that corrupts the communication.
  Take care that your cabinet is wired according EMC rules, you can find some tips at the end of this manual.
  If the USB chipset is slow, you could solve it by using a PCI USB add-on card. If this is not possible, you need another PC.

- Ethernet communication too slow. This could happen if the settings are not 100% equal to what is described in the setup. E.g. for the used adapter card only the TCPIP protocol must be on and all others must be OFF. There must be a 1:1 connection from Controller Board to the PC using a CROSS CAT5E cable. So you cannot connect the CPU as part of your home network, it must be on a separate network adapter.

  Theoretically also Ethernet may suffer from EMI disturbances due to bad unshielded cabling. In practice Ethernet is very robust to this. Anyway always make the wiring with the EMC rules in mind. See hardware tips at the end of the manual.

  There is also this experience, heavy browsing on internet while doing CNC may cause FIFO UNDERRUN error, especially on Windows XP. Windows 7 and Windows 8 are a lot better than Windows XP with this.

G0 Feed Factor:

With this you can apply a factor for the feed used at G0, this allows different feeds for G0 positioning G1/G2/G3 milling.

G0 Acc Factor:

With this you can apply a factor for the acceleration used at G0, this allows different acceleration for G0 positioning G1/G2/G3 milling.

FeedOverride input:

You can select "UI", "UI & Hand wheel" (Default) or "analogue input 1 – 3" on the CPU, with analog input you can use a potentiometer to control the feed override, recommended potentiometer value is 4K7. When UI and hand wheel is selected you can control the Feed Override using the F+ and F- buttons and the Hand wheel to control the feed Override from 0-300%. Note that the machine will not go faster than the maximum velocities of the motors allow.

FeedHold input: Here you can select a digital input from the CPU that sets the Feed Override to zero immediately when activated. When released the Feed Override will go back to the value before. This function may be used by EDM machines to stop the feed when there is a short circuit detection of the electrode.

2.1.7 Kinematic Setup

Trivial kinematics:

It is not needed for normal Cartesian machines, leave the Trivial 1:1 kinematics checked. Please contact Eding CNC if you have a special machine or robot with non-Cartesian axes.

2.1.8 Tool change Area setup

XYZ Limits: By setting the limits here to a value different from zero, the TCA (Tool Change Area) guard will be activated. Using the values here you define an area on the machine which is restricted to tool change. A normal work piece program is not allowed to enter this area.
You can also not jog in this area or move to this area by MDI.
If you need to be in this are issue command "TCAGuard off" To re-enable the protection "TCAGuard on".

Z DownToolLength:

For machine configurations where the tool chuck does not touch the machine bed when the machine is at its lowest Z position. Here you specify the tool length of the tool that fits when Z is at its lowest position.
This information is important for collision guarding.

2.1.9 Tangential knife setup

TanKnife Angle:

Tangential Knife is a rotation motor (the C Axis) around Z. Tangential Knife works

with normal G1, G2, G3 without tool-radius compensation G41, G42. The knife is rotated automatically in the direction of the X-Y move. This parameter determines the angle which 2 lines/Arcs can make without lifting the Z. If the angle is greater as this value, the Z will move up (G0), rotate the knife (G0), then move down again (G1). If the angle is lower, the rotation will take place without moving Z up.

TanKnife Z up distance:
Specifies the distance to lift up Z when detected angle is greater than Tan Knife Angle.

TanKnife blend angle and blend distance:
When subsequent lines have an angle with current line which is less than the blend angle and when the subsequent line or arc length is less than specified length the knife is not rotated before the move but during the move. For small angles in combination with small segment lengths this is in practice tolerable and will speed up the cutting process a lot. Be aware however that the knife direction is not exactly in the cutting direction and my break if the angle is too large.

The tangential knife is switched on and configured by interpreter commands.
These commands can be typed in MDI, part of the G-Code or hooked under the user buttons in macro.cnc.

;Switch tan knife on or off. Tan knife must first be configured in the setup.
Tanknife off
Tanknife on

;map tan knife to B or C or both axis output.
Tanknife b
Tanknife c
Tanknife bc

Especially for BEND tangential knife
Tanknife lo <lo-position>
Define the low Z value in work coordinates, this is the deepest point in the groove (*).

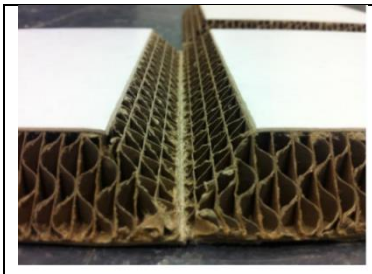
Tanknife hi <hi-position>
Define the high Z position in work coordinates, usually is 1-5mm above the material (*).

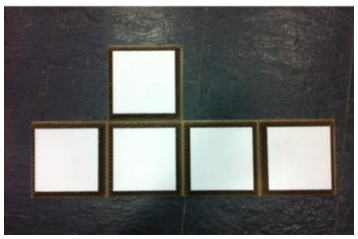
Tanknife bend -1 | 1 define the direction to which the knife is bend for a 45 degree knife (*).

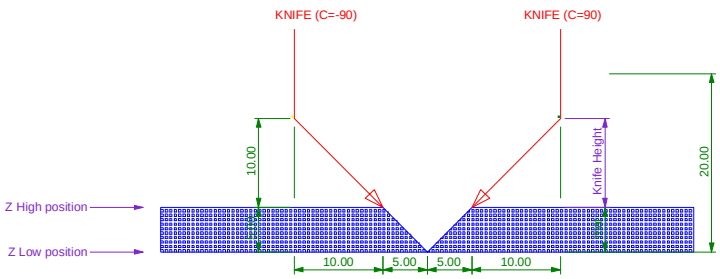
Tanknife rw <nr of turns> knife will be rotated back after nr of turns (anti-windup of wires). (*).

(*) These parameters are saved and used next time with **Tanknife on** command.

For usage of tangential knife that is bend 45 degree do calibration of the hi and lo position first. Then switch the knife on by tanknife bend -1 or +1 depending on the side to which it is bend. Set the C position to 0 when the sharp edge is pointing straight to the +X direction.







It is clear that with the bend knife during a z-move, the XY axes should move as well with same distance as Z, in the direction such that the sharp side of the knife moves under 45 degrees into the material. The user will program Z moves, the interpreter will rotate the knife (C axis) into the correct XY direction. Then kinematics will add the XY correction depending on the Z position.

The correction direction is the C axis angle + or - 90 degrees depending to which side the knife is bend. The correction amount in XY is the same as the height from the deepest point in the groove to the height where the knife is bend. So, if we define the tan knife low position as the deepest pointing the groove, then the compensation distance can be calculated as: CD = Z current - Z low + knife height.

The low position is where the XY correction is zero. This is the Z position at the bottom of the V-groove. The high position is where the compensation starts when coming from above. This should be at the surface of the material or little above.

2.1.10 Safety/Door open Input setup

Safety Input Selection: Select one of the AUX inputs to act as safety input, when active, only low speeds are possible and the running g code goes to pause (Feed hold, spindle off). This can only be configured for CPU5B.

Safety Feed: Feed in [mm/s] to be applied when the safety input is active and when the machine is not homed and homing is mandatory is set.

Intended use is for a door switch.

2.1.11 Spindle and PWM setup

MinS: The lowest possible speed for you spindle. If a command for a lower S values is used, then this minimum value is applied.

MaxS: The speed of your PWM controlled spindle when the PWM signal is at 100%.

Ramp up Time: The software waits this time between switching on the spindle and starting the further machining.

StopOnPause: Stop spindle when pause is activated.

zUpOnPause: Z goes automatically up when pause is activated.

autoStartAfterPause: When start is pressed, the X,Y axes automatically reposition to the pause position, the spindle is started, then the Z goes down with approach feed and the program continues.

zUpDistance: Hom much the Z goes up when when pause is activated and zUpOnPause is active. Software will protect against going up beyond the Z limit.

approachFeed: The feed used for Z down when autoStartAfterPause is on and the start button is pressed.

RPMSensor: Check if you have connected a spindle speed sensor to the Sync input of the CPU. The sensor should give 1 pulse/revolution, minimum pulse width 0.5ms.

MistIsSpindleDirection: Special for CPU5A, use mist output for spindle direction if you need it (CPU5A has no separate spindle direction output, that is why).

IsStepperMotor: Step/Direction pulses will be generated if set instead of PWM. A Stepper motor or Servo Spindle can be connected to the PWM/TOOL-DIR output when selected.

SmoothStep: Only in combination with IsStepperMotor, when checked the ramp-up-profile is smoother than when not checked, however when checked PWM2/PWM3 on the CPU5B cannot be used as separate PWM output anymore..

CountsPerRev: Provide the number of steps/revolution here if IsStepperMotor is checked.

SpeedOverrideInput: Specify UI or analogue input for controlling the speed (CPU5B).

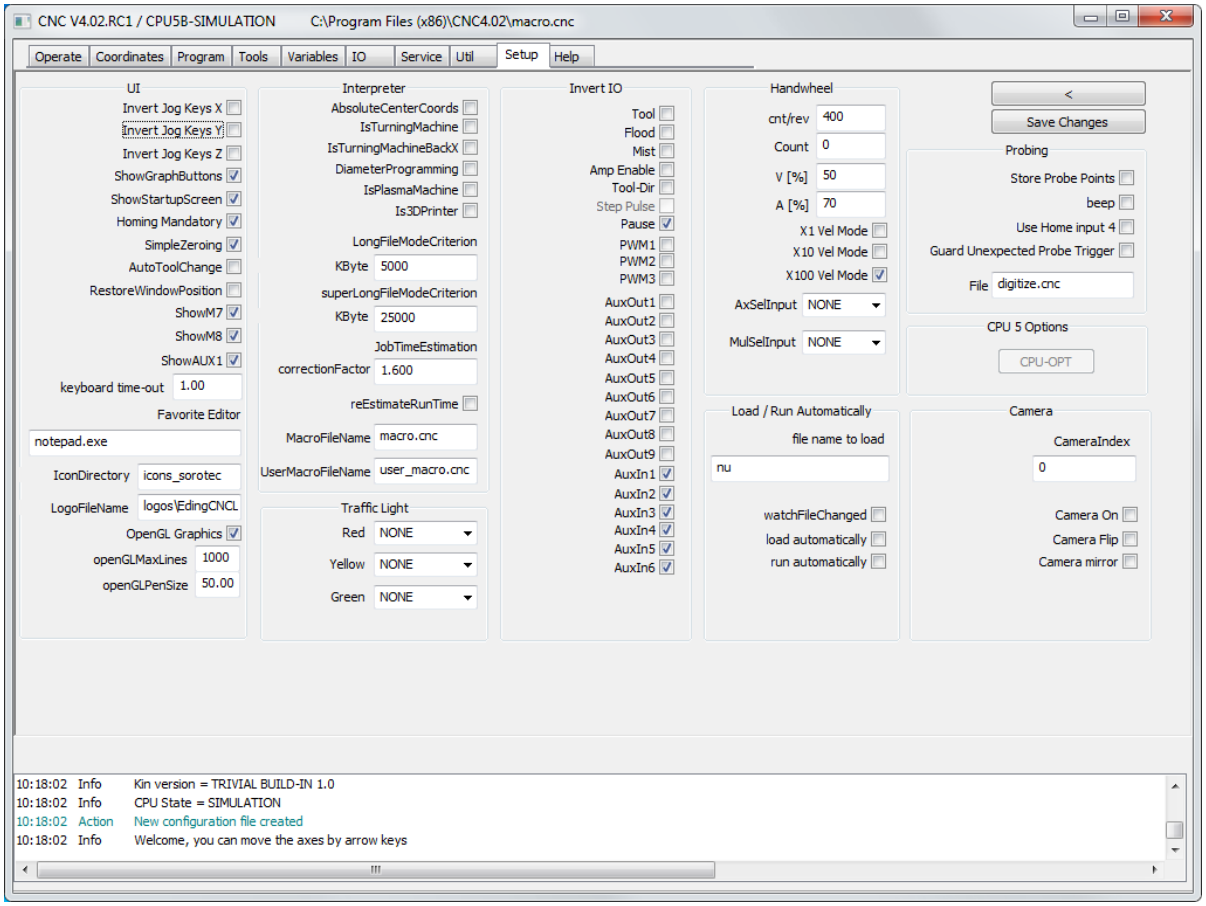
SpeedHoldInput: Specify digital input for speed hold, when activated spindle speed goes to zero. When release spindle restarts. Some applications need this functionality.

SpindleReadyPortID: User definable input which indicates that the spindle has reached it speed. It is currently only available by editing the cnc.ini file.

showInProgSpeed: This setting is under [USERINTERFACE] in the cnc.ini file.
It can have these values:
0: Show programmed speed (Default)
1: Show PWM value
2: Show analog in 1
3: Show analog in 2
4: Show analog in 3
hese options are convenient if the spindle has an analog output for power measurement.

showInProgSpeedAnaMulFactor :
Multiplcation factor for the analog value for option 2-4 with **showInProgSpeed.**

Setup Page 2, press ">" button on first setup page to get here:



2.1.12 **UI setup items**

Invert JogKeys: Inverts the movement of the keyboard keys, for moving bed machines, the bed moves in the direction you press the arrow.

IsTurningMachine: Check if your machine is a Lathe, this effects mainly the 3D display which shows the X-Z plane for turning. Also the jog keys operate differently. Futher the working plane is set to G18 (X-Z).

ShowStartupScreen: When checked, the startup screen is shown when EDINGCNC starts.

HomingMandatory: When checked, running a job and mdi is not allowed before the machine is homed. Also the jog speed is limited to 5% speed. This feature prevents damage to your machine because when the machine isn't homed, the limit guards are not working. So, I advise to leave this item checked always.

SimpleZeroing: If checked, the zero buttons (beside the position display), will simply set the work position to zero. If this item is not checked, a dialog will be shown in which you can set the position. Default it shows a value which is – tool-radius of the current tool. This is handy when zeroing from the lower left corner with the endmill against the material.

AutoToolChange: If checked the running job will not stop when a toolchange is encountered. Use this when you have a ATC or if you simply always have the tool already in.

ShowMaximized: GUI will start maximized taking the whole screen.

ShowM7: Show or hide M7 button

ShowM8: Show or hide M8 button

ShowAUX1: Show or hide AUX1 button

KeyboardTimeout: Time out is used while jogging. This feature is introduced because of the use of Bluetooth keyboards. It could happen that jog start is pressed, but never released (jog stop) because to keyboard is no longer in reach. The timeout will automatically jog stop of needed. The default value of 1s is OK for most PC's. I do not recommend to set it lower because you may get unwanted time-outs.

REMARK: More ShowXXX options are available under [USERINTERFACE] in the configuration file cnc.ini.

ShutDownOnFatal: If checked software will shutdown automatically when a fatal error such as disconnected CPU occurs. This may be used when the electrical power is switched of and connection to the CPU gets lost.

Favorite Editor: Specify your favorite editor here. I recommend notepad++, it is freely downloadable at internet. E.g. for notepad++, specify c:\program files\notepad++\notepad++.exe.
The advantage of notepad++ is that the editor jumps to the actual G-Code line immediately, very handy when programming G-Code.

IconDirectory: The name of the directory where the GUI icons are located.
nu means **n**ot **u**sed.
If you want to change the Icons on the buttons, you can make first a copy of the entire **icons** and name that directory to **myIcons**.
Make your changes an place the directory name in this field.

OpenGL: Check to use OpenGL graphics. This allows smooth panning, zooming and rotation using the mouse.
Left mouse key; **Pan**
Right mouse key: **Zoom**
Control+Left mouse key: **Rotate**.

OpenGLPenSize: Set PEN size shown in graphic, size is in millimeter.

2.1.13 IO setup

Invert IO: Check if you want to invert the output.

2.1.14 Interpreter settings

DiameterProgramming:
Check if you want diameter programming for turning, all X-axis values are interpreted as diameter. The effect is that all movements in the X-axis are divided by 2.

AbsoluteCenterCoords:
If Checked, the I,J,K value is interpreted as absolute value. Incremental is used mostly.

LongFileModeCriterion:
Specify a number of Kbytes here. When the loaded job file is larger, the UI switches to long file mode. The program listbox changes and the graphics will show only outlines when a program is loaded. This is al needed to preserve memory and speed for large files.
In this mode the file itself is still executed from memory and allows complex G-Code constructs (While, If then else, sub routines).

SuperLongFileModeCriterion:
Specify a number of KBytes here where super long file mode starts. This number should be equal or bigger as LongFileModeCriterion. For very long files from 20MByte and UP to 4G this mode is required. It also puts the GUI in the same mode as with LongFileMode, but as extra, the file itself is no longer executed from memory. The means that complex G-Code constructs are no longer possible. These type of files contain only straight forward g-code without "while-endwhile", "if-then-else" and "subroutines". The tool changes are still executed from the macro.cnc file, so full automatic tool change is still available.
Files with up to 100.000.000 lines of G-Code have been tested with this.

Macro Filename: Name of the macro file, it can be changed, the default is macro.cnc.

User Macro Filename: Name of the macro file, it can be changed, the default is macro.cnc.

Note that this 2nd macro file called user macro file is new since V4.02. The intention is that the user puts his own macro's in this file and that the Macro File contains system things that can be changed by EdingCNC with new versions. Eding CNC will not touch the User macro file.

2.1.15 Traffic light setup

Red: Specify output for RED color.

Yellow: Specify output for YELLOW color.

Green: Specify output for GREEN color.

CPU5B is required to view all colors because other CPU's do not have enough amount of outputs.

2.1.16 **JobTimeEstimation**

During the Render phase, after loading the job, the job time is estimated. But this is just a quick estimation because a real calculation of time would take too much time, therefore these parameters:

CorrectionFactor: Correction factor for the time calculations, you can change this if you see that your type of jobs require a correction.

RestimateRunTime: When checked, you will see the remaining estimated time of job based on the average speed measured and the total distance to go.

2.1.17 **Hand wheel Setup**

Cnt/Rev: The number of counts of the hand wheel for one revolution, usually 400 for most CNC hand wheels.

Count: Shows the actual Hand wheel count value, try to turn the hand wheel and see it change.

V[%]: Percentage of velocity from selected axis, this is the maximum **velocity** the axis will move when using the hand wheel.

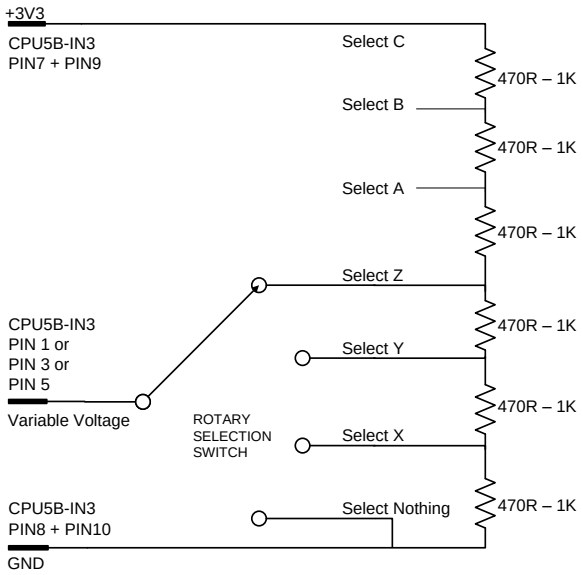
A[%]: Percentage of acceleration from selected axis, this is the maximum **acceleration** the axis will move when using the hand wheel..

X1..X100 Vel Mode: In velocity mode the most important is that the movement stops immediately when the rotation of the hand wheel stops. The position of the hand wheel will not be maintained if velocity mode is on. The position of the handheld is maintained if velocity mode is off. This also means that the axis may not immediately stop if the hand wheel rotation stops. When turning beyond the limits of the axis, you have to turn back the hand wheel the same amount before the axis starts moving again.
My own experience is that it works best to use velocity mode at X100 only. Jus play with it to experience the behavior and make your own choice.

AxSelInput: Specify analogue input to be used for axis selection. A multi switch with 5 1K resistors can be used to make this:
See hardware specification CPU_5B_FLYER_TECH.PDF, it is on the download page of the website. This option is only applicable to CPU5B. The max analog input value is 1023 and this corresponds with 3.3 Volt.

INPUT VALUE	**VOLTS**	**AXIS SELECTED**
< 100	<0.32V (0.00)	-
120-220	0.38 – 0.71 (0.55)	1
290-390	0.94 – 1.26 (1.10)	2
460-560	1.48 – 1.81 (1.65)	3
630-730	2.03 – 2.35 (2.20)	4
800-900	2.58 – 2.90 (2.75)	5
> 970	>3.13 (3.30)	6

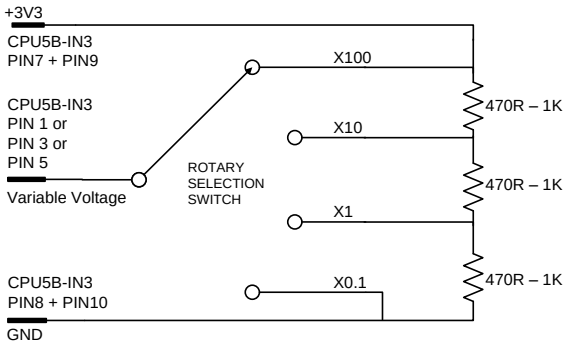
You can achieve this by putting 6 identical resistors in series and a rotation switch. Example made for OFF-XYZ axis:



MulSelInput: Specify analogue input to be used for multiplication factor selection X0.1 x1 x10 x100. This option is only Applicable to CPU5B.

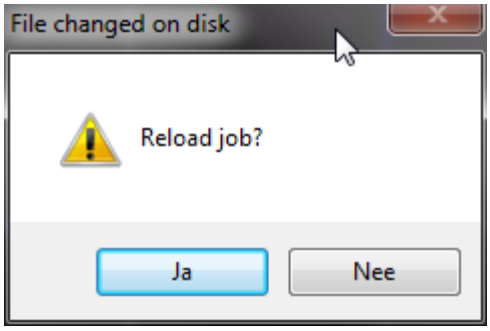
INPUT	VOLTS	MUL. FACTOR
< 100	<0.32 (0.00)	0.1
291-391	0.94 – 1.26 (1.10)	1
632-732	2.04 – 2.36 (2.20)	10
> 900	>2.9 (3.30)	100

You can achieve this by putting 3 identical resistors in series and a rotation switch:



2.1.18 **Load/Run Automatically**

watchFileChanged: If checked EDINGCNC will watch the loaded g-code file for changes on disk if EDINGCNC is not running. When it is changed, e.g. by an editor or because it is saved by a CAM software, then EDINGCNC will ask you to reload the file:



load automatically: If this is checked, the file is automatically loaded when it changes on disk, no dialog will appear.

run automatically: If this is checked and also the load automatically check, then the file will be loaded and immediately start running when changed on disk.

fileName: This is the name of the file that EDINGCNC watches at startup. So if EDINGCNC is started and this file time/date changes on disk, it will be loaded.
If manually another g-code file is loaded, then USNCNC will watch that one.

2.1.19 **Probing Setup**

StoreProbePoints: The touch points are stored in a file when this is checked. This is used for digitizing.

Use Home input 4: If checked home input 4 is used instead of the standard probe input.

File: The file name for storing the touch points. The file is opened at the first probe touch en closed when a M30 command is encountered, usually at the end of the G-Code program.

M95: Using a probe which is fixed on the machine. In this case the probe-tip will have an offset. For this you can use the M90-M97 function. With this it is possible to select an offset for Spindle 1 (M90), Spindle 2 (M91), spindle 3 (M92), probe (M95) or camera (M97).

The offset can be calibrated as follows:

1. Take care that M90 is active (1st spindle), if not execute m90 in MDI.
2. Mark a point on your machine bed and accurately move the **tool-tip** to this point.
3. Zero the axes X,Y,Z at this position.
4. Now move the **probe tip** exactly to this position.
5. Execute in MDI "M95 Q1".

2.1.20 **Camera Setup**

CameraIndex: Use 0 if you have only one camera, use 1 if you have 2 Cameras and want to use the 2nd one.

CameraOn: Select Camera if used.

CameraFlip: Flip image vertically

Camera mirror: Mirror camera image horizontally

Rotation: Rotate Camera image [Degrees]

Camera offset calibration, if you have mounted the camera on the machine the camera will show a different position as the position of the tool-tip. The difference is Camera offset.

The steps to calibrate this offset are:

1. Take care that M90 is active (1st spindle), if not execute m90 in MDI.
2. Mark a point on your machine bed and accurately move the **tool-tip** to this point. (Tip: drill hole at that point)
3. Zero the axes X,Y,Z at this position.
4. Now move the **camera** exactly to this position.
5. Execute in MDI "M97 Q1".

   Now when calibrated, you can use M97 to make camera offsets active.

   Tip: Use M97 P1 to position the camera at the tooltip position. Use M90 P1 to activate and move the main spindle to the work position.

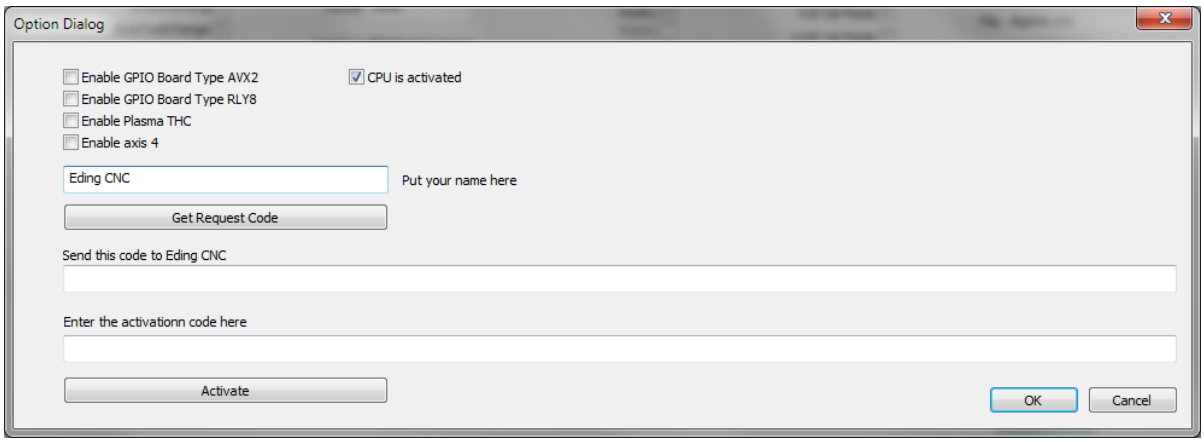
   In the GUI you can also select this by pressing control+F2 in the graphic menu. Just try and you will see it.

2.1.21 **CPUOPT**

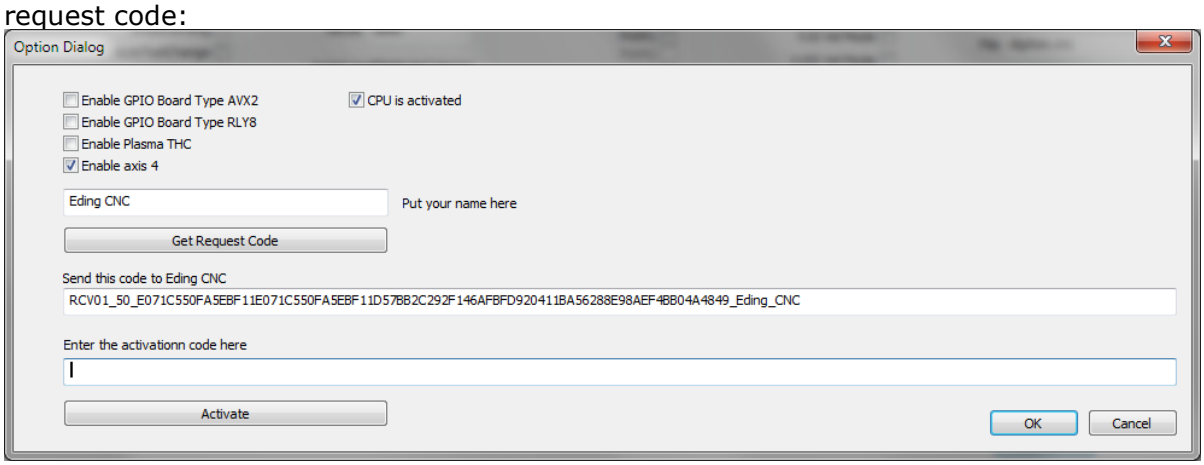
CPU and Optional functions can be activated here:

- Enable the new AVX2 IO board
- Enable the new RLY8 IO Board
- Enable Plasma Torch Height Control
- Enable the 4th axis on a CPU3A

Note: This activation is not needed if you have after you bought a CPU.
It is needed only to enable optional functions.



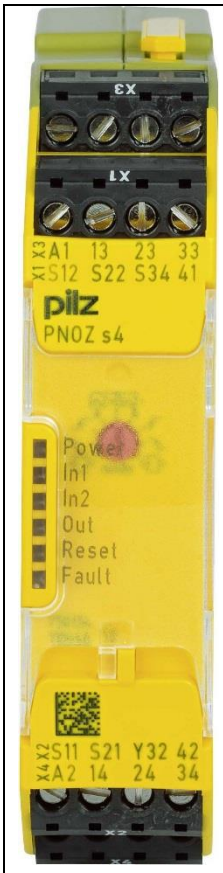
These are the steps to follow:
In the dialog check e.g. the "enable axis 4" checkbox, enter tour name and press get request code:



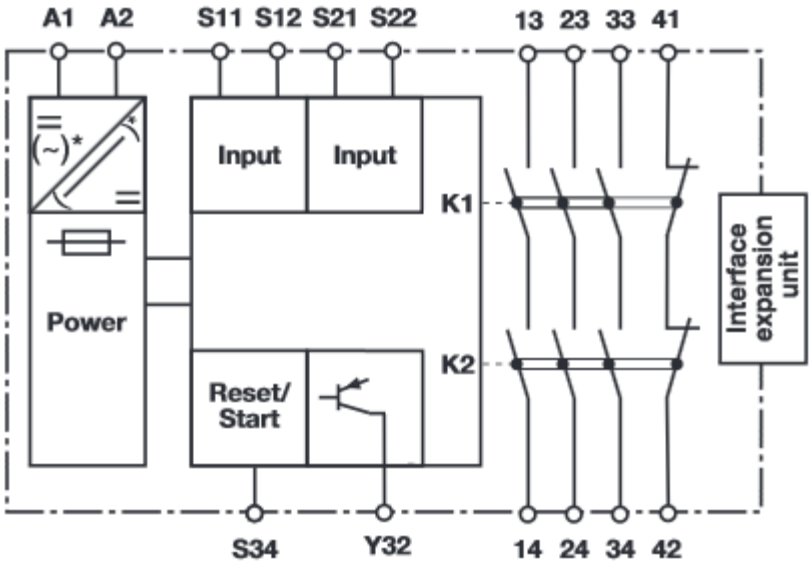
Send the request code to the supplier.
Copy and paste it into an email and send it to your EDINGCNC supplier.
To do this double click the code, press ctrl-c, in your e-mail press control-v.

Your supplier will send you a activation code.
Copy and paste this into the activation code area. then press activate.

2.1.22 **Using a safety relay**



PNOZ S4



Safety of the machine should be independent of the CNC software.
The purpose of the safety relay is to check safety inputs like ESTOP buttons and possibly door switches or fences and allow to switch ON only if everything is safe. If a ESTOP button is pressed or some other way the safety chain is violated, the safety relay switches off the dangerous power in the machine. For a milling machine this is the spindle and the servo/stepper motors.

2.1.22.1 POWERING THE SAFETY RELAY

A1/A2 is connected to our 24V supply, the same supply as we use for the CPU card. This power is switched on when the electronic cabinet is switched on.

2.1.22.2 INPUT CONTACTS OF THE SAFETY RELAY

The relay has **input contacts S11/S12 and S21/S22** to which all ESTOP buttons and door and fence switches are connected. We use ESTOP buttons with 2 contact like this:

In this chain of safety contacts, the contacts of an extra relay is added, this relay is switched on when the CNC CPU is ready (system ready). Ideally this system Ready should be derived from the CPU Watchdog signal. So that the system will switch off if

the connection with the CPU is lost or the software hangs. The iCNC600 CPU's have a system ready output, for CPU5B this ouyput is defined in the cnc.ini file with:

```
systemReadyOutPortID = 0 ;Standard system ready port, 1 - 8 for output AUX1-AUX8
```

2.1.22.3 OUTPUT CONTACTS OF THE SAFETY RELAIS

The relay has **output contacts 13/14**, **23/24**, **33/34** to switch ON/OFF power of safety related equipment. These output contacts are not intended to switch heavy loads, so they are used to drive external power relays that switch on the power for the spindle and drives. Addition: The machine guidelines require not 1 relay, but 2 in series to switch this power. This is because a relay can mall function, 2 relays in series are safer.

OUTPUT contact **13/14** will switch ON the SPINDLE POWER using 2 external power relays. Our frequency converter has a special enable input to which we connect the **13/14** output. OUTPUT contact 23/24 will switch ON the power supply for the motors using 2 external power relays in series. We use a the same standard DIN rail relay 2x. OUTPUT contact **33/34** will be connected to the ESTOP input of the CPU (ESTOP 1 for CPU5B).

Example of a DIN rail power relay.

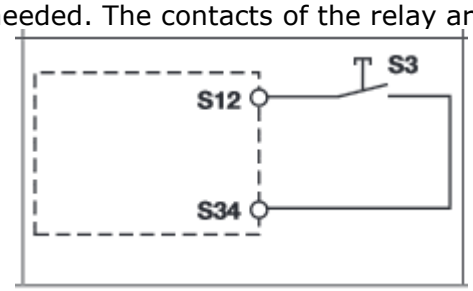
2.1.22.4 SWITCHING ON THE SAFETY RELAY

If everything is safe the relay can be switched on via a special RESET input.
This reset input can be a physical button or software operated. We want it to be software operated, using the MACHINE ON button in the UI:

We set the relay mode to operate at a positive edge on the S34 input:

Operating mode selector switch "mode"	Automatic/ manual start	Monitored start rising edge	Monitored start falling edge	Automatic start with start-up test
Without detection of shorts across contacts				
With detection of shorts across contacts				

We use the AUX1 output of the CPU to reset the safety relay and connect this output to the AUX 1 output of the CPU. The iCNC600 has 24V outputs and can directly be connected to **S34**. CPU5B has open collector outputs and cannot be directly connected, here a small 24V relay is needed. The contacts of the relay are connected like this:



Configuration of the safety relay is done in the settings file, cnc.ini.
These are the parameters, which are located under section

[SAFETY]
safetyRelayPresent = 0 ;Set to 1 if safety relay present
systemReadyOutPortID = 0 ;Standard system ready port or use 1-9 for AUX port 1-9
safetyRelayResetOutPortID = 0 ;1-9: AUX1-AUX9
safetyRelayResetDelayMs = 500
safetyRelayPulseLengthMs = 250

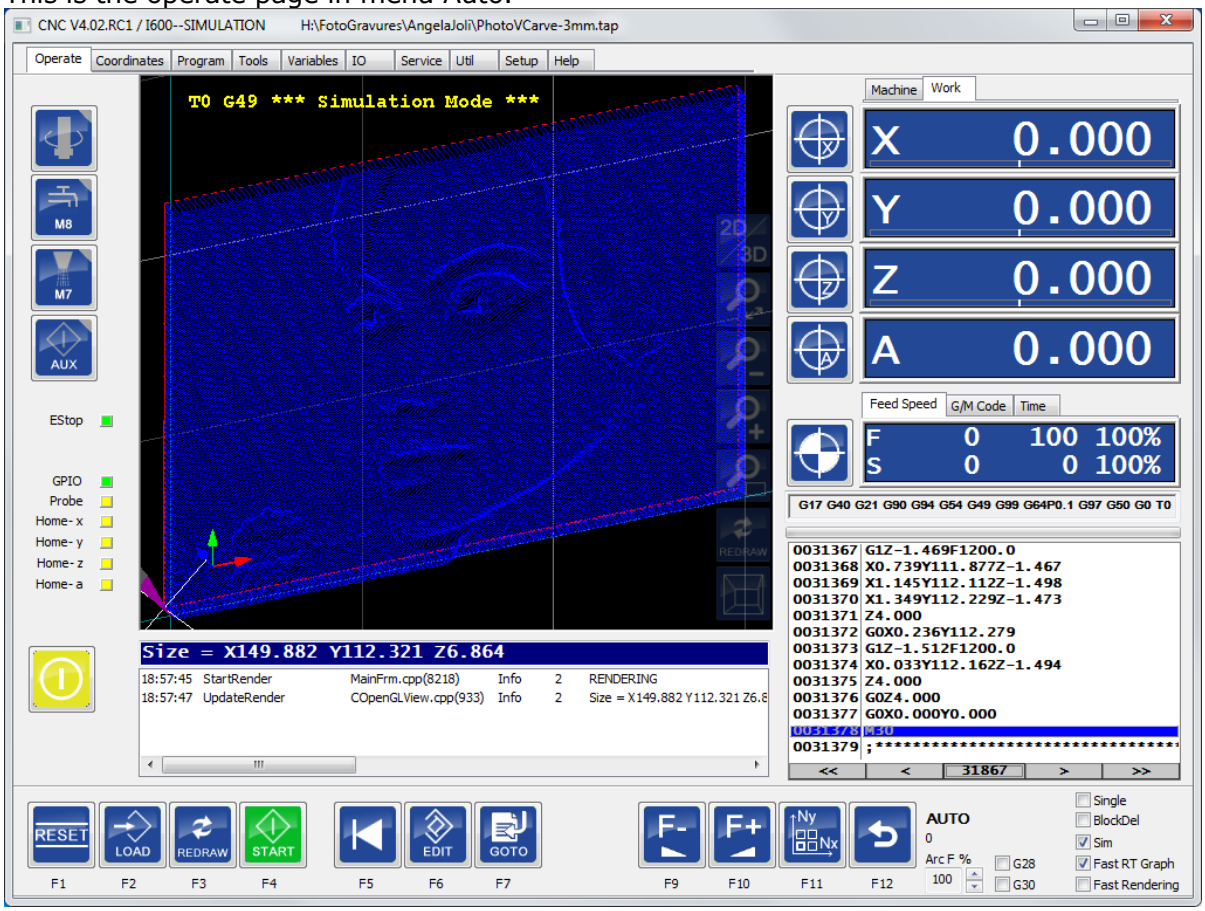
Now, the sequence to switch on is as follows:

- Make sure the ESTOPs are not pressed.
- Press the MACHON button in the UI The software will first switch on the **systemReadyPortID**, (note that this port switches a relay that is part of the safety chain) then wait 500ms and then generates a 250ms pulse on the **safetyResetOutPortID**. Finally the software waits again 500 ms and now checks id the ESTOP input has switched. Of yes, all is OK and the machine is ON.

- Next press the RESET button, this will switch on the amplifier enable.

2.2 OPERATE PAGE, THIS IS WHERE THE MACHINE IS OPERATED FOR MILLING

This is the operate page in menu Auto.



2.2.1 Operate page introduction

From this screen all machine operation like jogging, running a job, etc can be executed. The Operate screen is designed such that it is mouse, mouse-less and touch-screen friendly.

In the middle we see the graphics showing the tool path. Blue/Red when loaded and rendered. Yellow/Green when actually running. So it shows the tool path real-time.

At the left side there are buttons for common used IO:

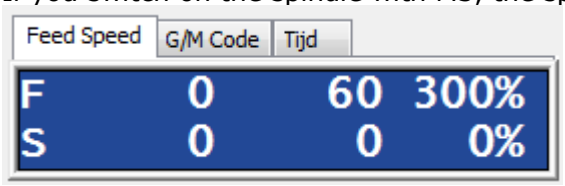
- Spindle on-off,
- Flood
- Mist Coolant on-off, and
- AUX on-off (e.g. for the machine light).
- MACHINE ON Button (Below Home C led)
  This one has a few colors with different meaning:
  - Grey means machine is off, drives switched off.
  - Yellow Flash, amplifiers must be enabled, homing must be performed.
  - Yellow, waiting for operator action.
  - Green, machine running.
  - Red, error or estop. Flash when E-Stop still active.

The right part of the screen shows the axes positions, when homing you use the **machine coordinates** and for all other operations the **work coordinates**.

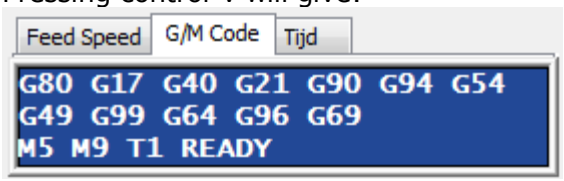
The buttons beside the axes positions are for zeroing the work position, on the background a G92 command is executed to perform this. The zero buttons can also be found in the zero submenu, especially for people who do not like using the mouse at the machine.

below the machine positions we see the general status window:
You can select FS (Feed Speed), GMT (G-Code, M-Code, Tool) and T (time estimation for running job).
There is a shortcut key: ctrl-v to change the selection here.

Feed/Speed
You see the actual value, set value and percentage.
If you do a G1 in this example, the feed will be 60.
If you switch on the spindle with M3, the spindle speed will be set to 100 rev/minute.

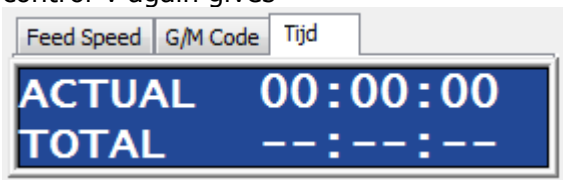


Pressing control-v will give:



This shows the actual G code and M code status as well as the actual tool number and the machine state, READY, RUNNING etc.

control-v again gives



Here you see the actual running time of a job and also the estimated TOTAL time.

2.2.2 Reset Button F1

This button has to be used after starting the software to enable the drives.
The amplifiers are switched on when pressing the **reset button.** Try this, you can feel at the motor shaft if the amplifier is on. if you can still turn the motor by hand, you probably need to reverse the amplifier enable polarity in the setup.

But the reset button does more:

- Enable the amplifier
- Recover from Error after you get one
- Stop a running program

2.2.3 Escape Button

This button pauses the current job execution if running.
This is just there for convenience, not for safety emergency stop.
For safety use a real E-STOP button!

2.2.4 The menu's

2.2.4.1 Main Menu

The Main menu looks like this and has a user selectable logo at the right:

- F1 reset, this key comes back un every sub menu
- F2, to home menu.
- F3, to zero menu.
- F4, to auto menu.
- F6, manual data input (ctrl-f6 works always too for MDI)
- F7, machine I/O functions for spindle and coolants.
- F8, graphic manipulation functions.
- F9, jog with keyboard or hand wheel mode.
- F10, jog pad for jogging by mouse or touch screen.
- F11, user menu

2.2.4.2 Home menu

- F1, reset
- F2 - F7, Home X - Home C
- F8, Home all axes
- F10, go to g28 park position
- F11, go to g30 park position
- F12, return to main menu.

For homing setup see homing and coordinate systems chapter.

2.2.4.3 Zero menu

- F1, Reset
- F2 - F7, zero x - zero c
- F8, zero all
- F9, measure rotation and apply G68 R..
- F12, back to main menu

F9 measure rotation is a feature that makes life easy. It automatically corrects your work piece/clamp for rotation. This means that you no longer have to spend time to setup your clamp / material very accurately, EDINGCNC will automatically correct for you.

2.2.4.4 Auto menu

- F1, Reset
- F2, Load G-Code file
- F3, redraw (re-render whole program through g-code interpreter)
- F4, run/pause
- F5 rewind job
- F6 start editor
- F7 start a job somewhere given a line number. (e.g. after a tool breakage).
- F9, +Feed Override
- F10, - Feed Override
- F11, Show Nesting options.
- F12, back to main menu

G28: Perform G28 when the program finishes.
G30: Perform G30 when the program finishes.

ArcF %: Reduce Feed for large Arc's.

Single: Activate single step mode, when F4 (Start) is pressed, only 1 line of the job file is executed.

BlockDel: When active all lines with '/' in front will not be executed.

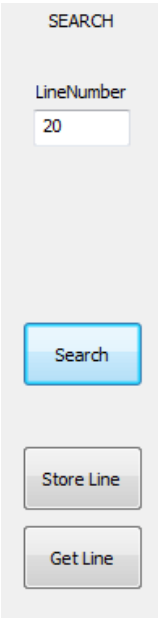
M1Stop: Optional stop M1, when an M1 is encountered in the g-code the program will halt if this check is on.

Sim: Simulation mode.

FastRtGraph:
The Realtime graph will not consume memory, use it when running long programs (several hours or more). This function is also automatically activated when the file size of the job is bigger than LongFileModeCriterion in the setup.

Fast Rendering: Also for very long programs, only the outlines (rectangle) of the part are drawn. This is also automatically activated when the files size is longer than SuperLongFileModeCriterion.

F7, set start line will give next popup dialog:

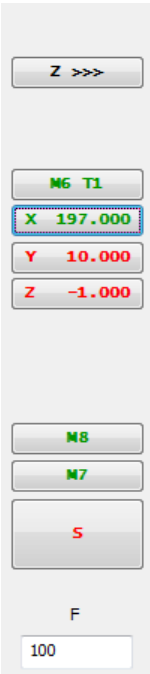


If you have stopped while Paused, the line number will show the current line of the job.
This happens also when you have pressed reset when paused.
Not that reset when pause is needed when you need to do e.g. a tool change.
During Pause only jog movements are allowed.

You can store and retrieve the stored line number using the Store/Get Stored buttons.

Press search to run the interpreter in Search mode up to the given line number.

The graphic shows the search.



When you press the RUN button (F4) after a search or pause, the following popup dialog may appear.
It appears only if any axis is not at the correct position or the spindle or coolants are not correct, it allows you to synchronize the actual situation with the required situation:

The Z >>> button will start move Z completely up.

The M6 T1 button shows the tool according to the interpreter,
This button is not visible at a start after Pause, only at start after search.
if the color is green the current tool matches the tool from the search status.
If the color is red, the tool doesn't match and you can start a tool change by pressing the button.

The Axis button show the position according to the interpreter on the searched line, green is match, red is no match, press the button to move the axis to the correct position, you can do this for all axes.
If any axis isn't synchronized, it will be done automatically when the start button is pressed.

The M8/M7 On buttons allow to switch on the Coolants.

The S button switches the spindle On with correct S value from the Search status.

F= Plunge rate is the feed rate for the movement towards the work piece, you can change this to a good value you want.

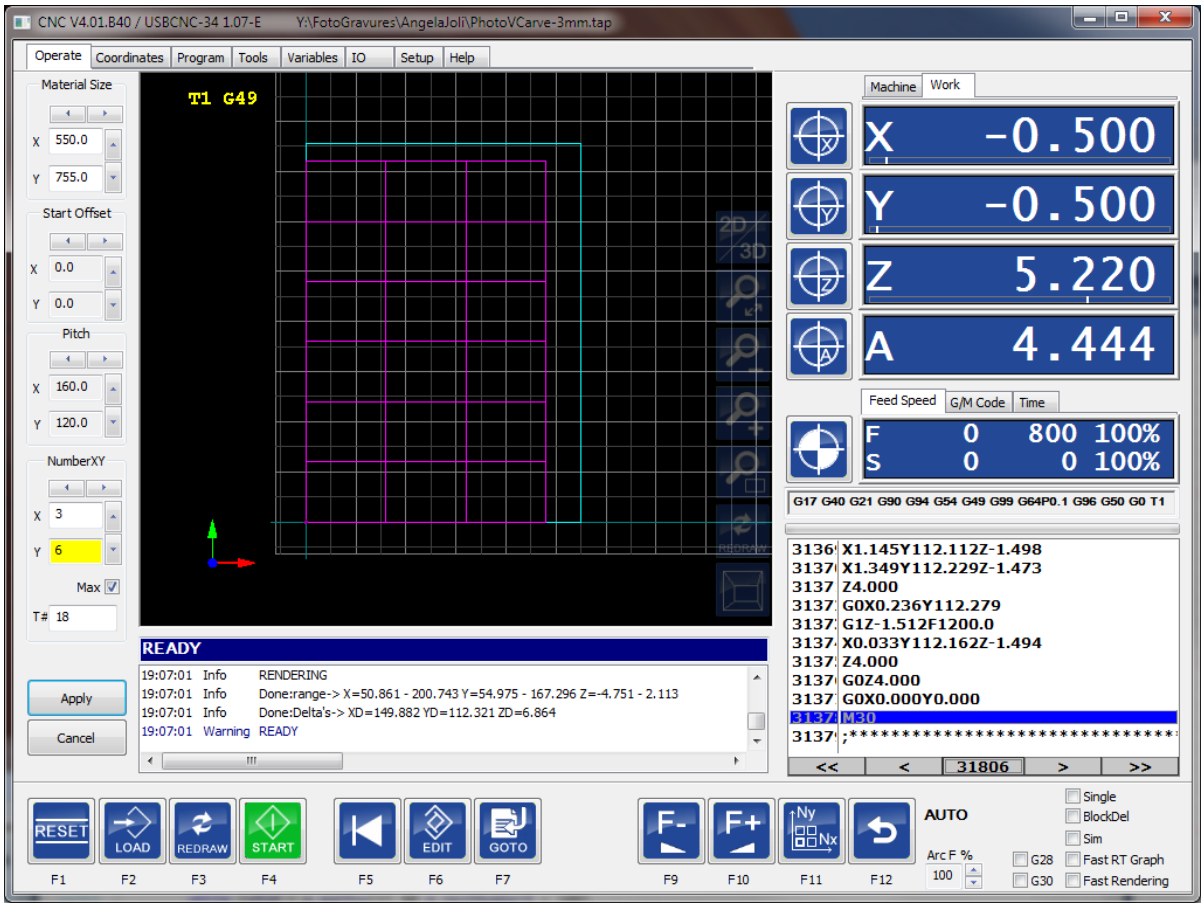
As last, the Run button, this will start a G1 with F towards the search positions, then restore the Feed to the search feed and start machining from there.
This way you are able to easily start half way in a g-code program.

For advance users of pause-start there are other options in the cnc.ini under [SAFETY]

```
;When 1 spindel stops on pause
stopSpindleOnPause = 1
;When 1 automatic start after pause occurs, movement and spindle is switched on.
autoStartAfterPause = 0
;Spindel goes up after pause
zUpOnPause = 0
;By this distance
zUpMoveDistanceOnPause = 10.0
;And this is the approach feed when auto starting in mm/sec.
approachFeed = 2.000000
```

F11, Nesting:

Nesting is a feature that allows to produce a product multiple times in X/Y ROWS:
Nesting is reachable if the machine is in READY state, you can always press RESET to get it in ready state if it isn't.



Material size: set the material size in X and Y, it is shown in the graph.
Start offset: set an offset for starting, play with it and you will see what it does.
Pitch: the distances in X,Y of the products.
Number: Specify the number of products.
Max: EDINGCNC will determine the max number of products.

Apply: Apply the current setting to the program.
Cancel: Cancel nesting, back to only one product.

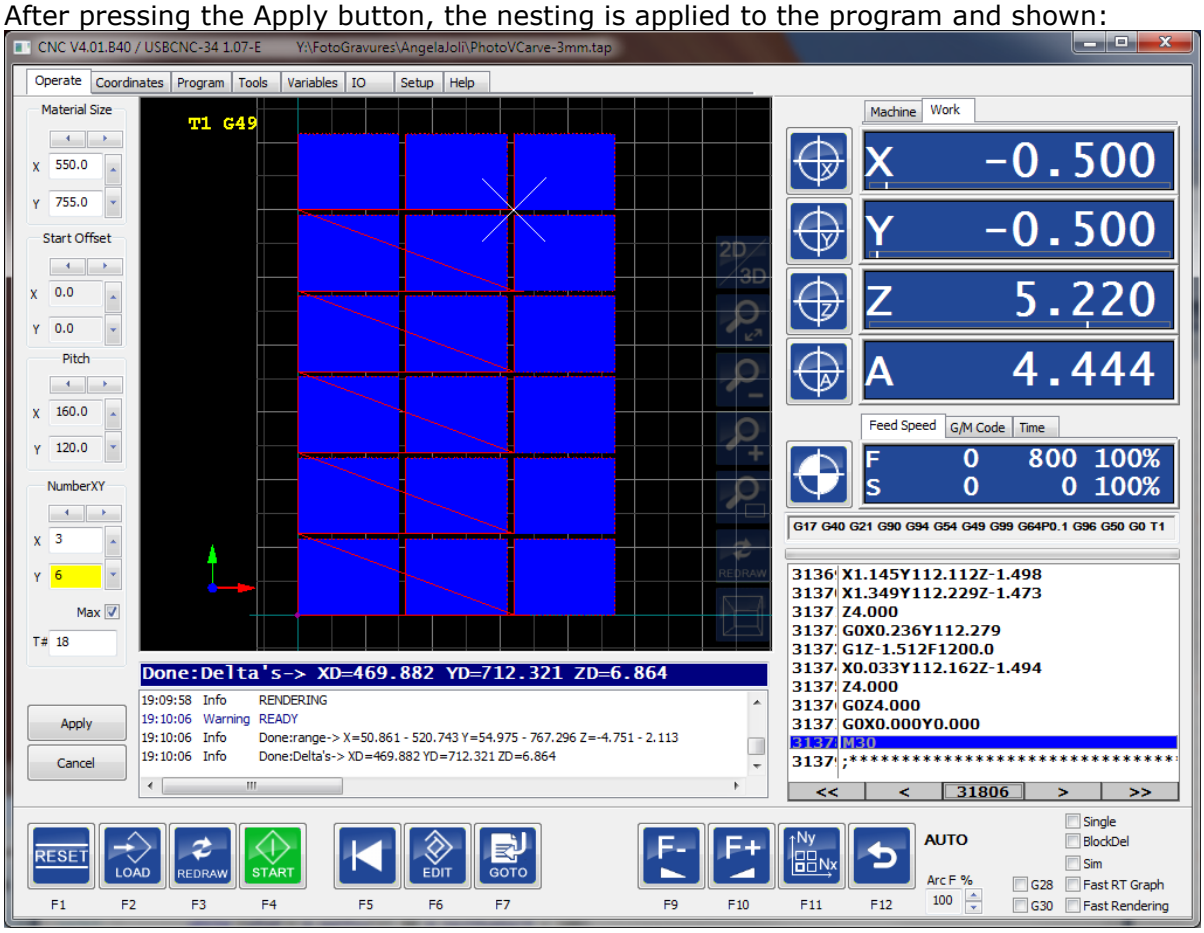
The Nest button F11, can be pressed to show/hide the nesting dialog.

Nesting internally uses coordinate system offset G59.3, the coordinate system offsets may not be used in the program, otherwise nesting will not work, so no G54 .. G59.3 allowed in the program.
G92 is allowed, but if changed must be set back to the original value at the end of the program.
The program must end with M30 otherwise nesting will not work.
Use M60 instead of M30 when the spindle should not stop between the work pieces.

The values above can also be set in the G-Code file like so:
%mx=200 Material size X
%my=200 Material size Y

%dx=200 the delta X or pitch X
%dy=200 the delta Y or pitch Y
%ox=200 the offset X
%oy=200 the offset Y

After pressing the Apply button, the nesting is applied to the program and shown:



Recommended is to create the g-code file for the product such that X0 Y0 is at the lower left side.

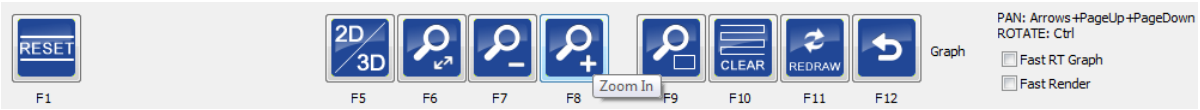
If you like to start not at the beginning, use the goto-line function and apply the NX NY values.

Happy production with Nesting!

2.2.4.5 IO MENU

- F1, Reset
- F2, drivers on/off
- F3, spindle on/off
- F4, spindle direction left/right
- F5, flood coolant on/off
- F6, mist coolant on/off
- F7, aux1 output on/off
- F9, Speed -
- F10, Speed +
- F12, back to main menu

2.2.4.6 GRAPHIC MENU



- F1, reset
- F5, switch between 2D X/Y plane and 3D iso-metric view.
- F6, zoom fit
- F7, zoom out
- F8, zoom in
- F9, zoom machine
- F10, clear
- F11, redraw (re-render whole program through interpreter)

The graph view shows a grid of **50mm** in mm mode or **2 Inch** in inch mode projected on the machine bed (X-Y surface). For a representative view it is important that the axes limits are correctly filled in and that the machine is homed manually or automatic. The current work coordinate system origin is shown as a cyan colored cross in the x-y plane. When you press the preview update button, a preview is shown of the loaded G-Code program. The preview is created by running the entire g-code file through the interpreter. So when interpreter errors occur, it shows in the log window and in the operate view the program list box shows the wrong line in red color. Note that there can be inaccuracy in what the display shows, this is there because of performance and memory usage limitation reasons.

Zooming, rotate, pan, 2D/3D view and other possibilities are found in the graph sub-menu, see the example below.

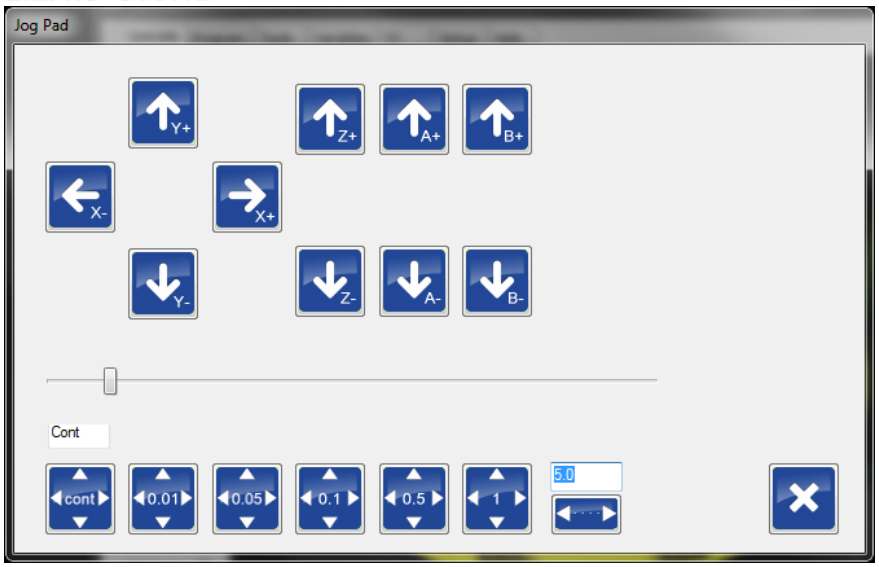
With OpenGL activated in the setup, real-time pan, rotate and zoom is possible with the mouse also:

pan : left mouse button
rotate : left mouse button + control
zoom : right mouse button.

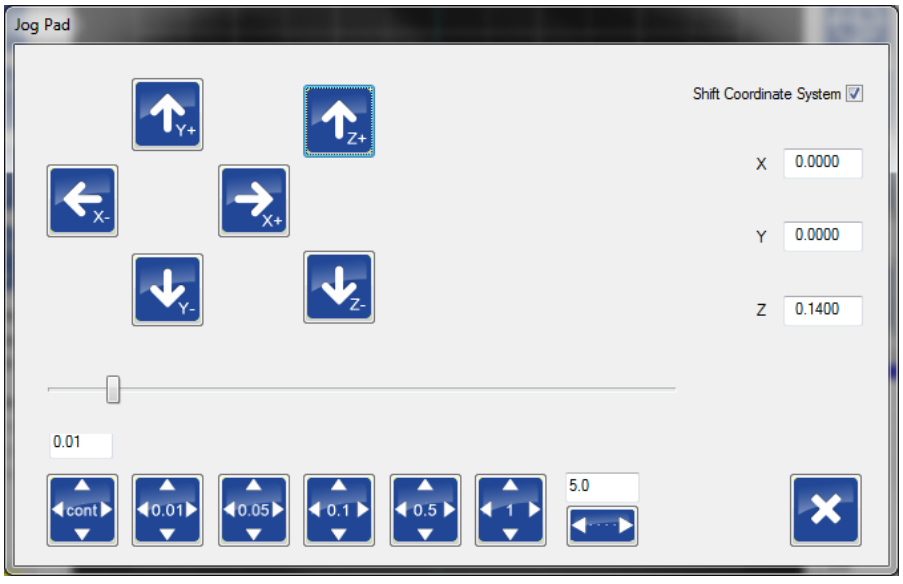
2.2.4.7 JOG MENU

- F1, Reset
- F2, jog mode continuous
- F3, jog mode step 0.0001 (Only visible in INCH mode G20)
- F4, jog mode step 0.001
- F5, jog mode step 0.01
- F6, jog mode step 0.1
- F7, jog mode step 1
- F8, jog mode step user value
- F9, jog mode hand wheel / mpg X1
- F10, jog mode hand wheel / mpg X10
- F11, jog mode hand wheel / mpg X100
- F12, return to main menu

2.2.4.8 JOG PAD



Jog by mouse, F12 return to main menu.
The function is similar to the jog menu, but it has some extra functionality with jog step.



When "Shift Coordinate System" is checked, jog-step functions as normal, the axes move one step at a time. The work position however remains the same. This is accomplished by modifying the active G92 offset. It is useful when e.g. during engraving you want to run the G-Code program again, but a little deeper in Z. E.g. you want to run the program 0.1 mm deeper, select jog step 0.1 and check "shift coordinate system" . Now press de arrow down button to move Z 0.1 mm down. Notice that the axis moves down but that the position remains the same. When you run your engraving program again the engraving will be 0.1 mm deeper into the material.

This option is also very handy during turning. Your program has run and you measure the work piece and see its diameter is still a bit too big. So now use the –X button to compensate the diameter. Run the program again and your work-piece diameter will be correct.

The amount of shift is shown at the right side. To reset the value to 0, which has no influence on the active offset nor machine position, uncheck, and then check "shift coordinate system".

2.2.4.9 USER MENU

- F1, Reset
- F2, Zero the Z coordinate using a flexible tool setter positioned on top of the material, see ZERO TOOL MACRO chapter.
- F3, measure the tool length and put the length in the tool-table using a fixed tool setter, see TOOL MEASUREMENT MACRO chapter.
- F4 - F11, user function user_3 .. user_10, user defined functions in macro.cnc
- F12, return to main menu.

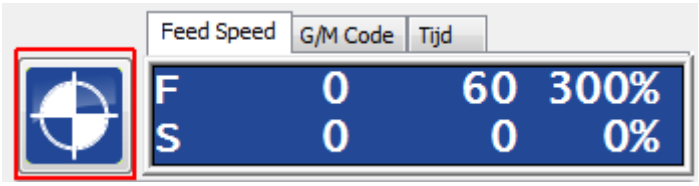
2.2.5 Operate page tasks

2.2.5.1 STARTUP

When you just started the application you have to press reset F1.
This will enable the drives, the machine on button left will be green flashing, this means the machine is ready but must be homed first.

2.2.5.2 HOMING

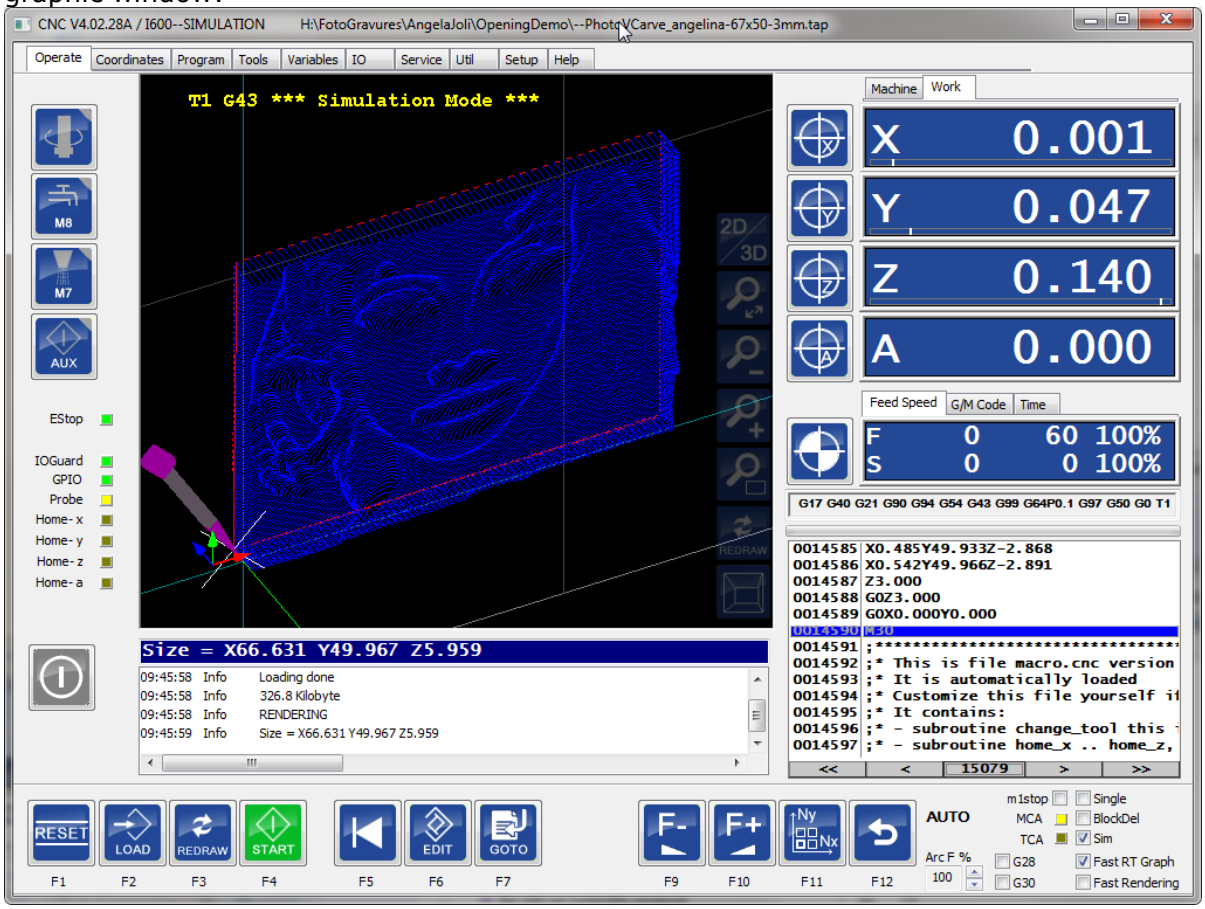
Homing is the next step to perform, this can be done via main->f2.
There you can do individual axis homing or home all axes art once.
For homing setup see homing and coordinate systems chapter.
All axes home at once can also be done using ctrl-h or the home all button beside the status:



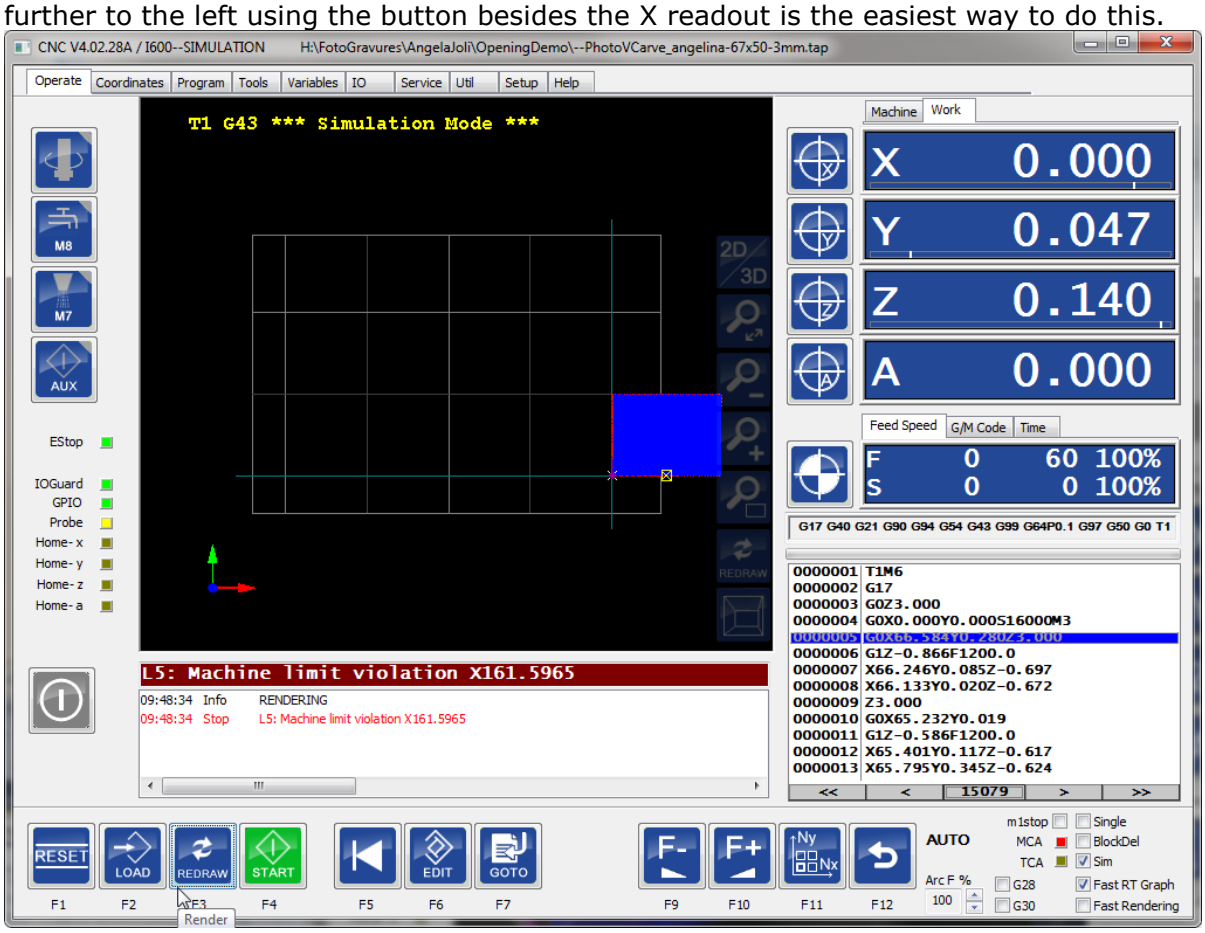
2.2.5.3 LOAD AND RUN A G-CODE FILE

After homing we a ready to run a program, we have to load a g-code file for doing that.
From the main menu press F4 (Auto), then F2 (load g-code file).
Go to the cnc-jobs directory and load demo.cnc.

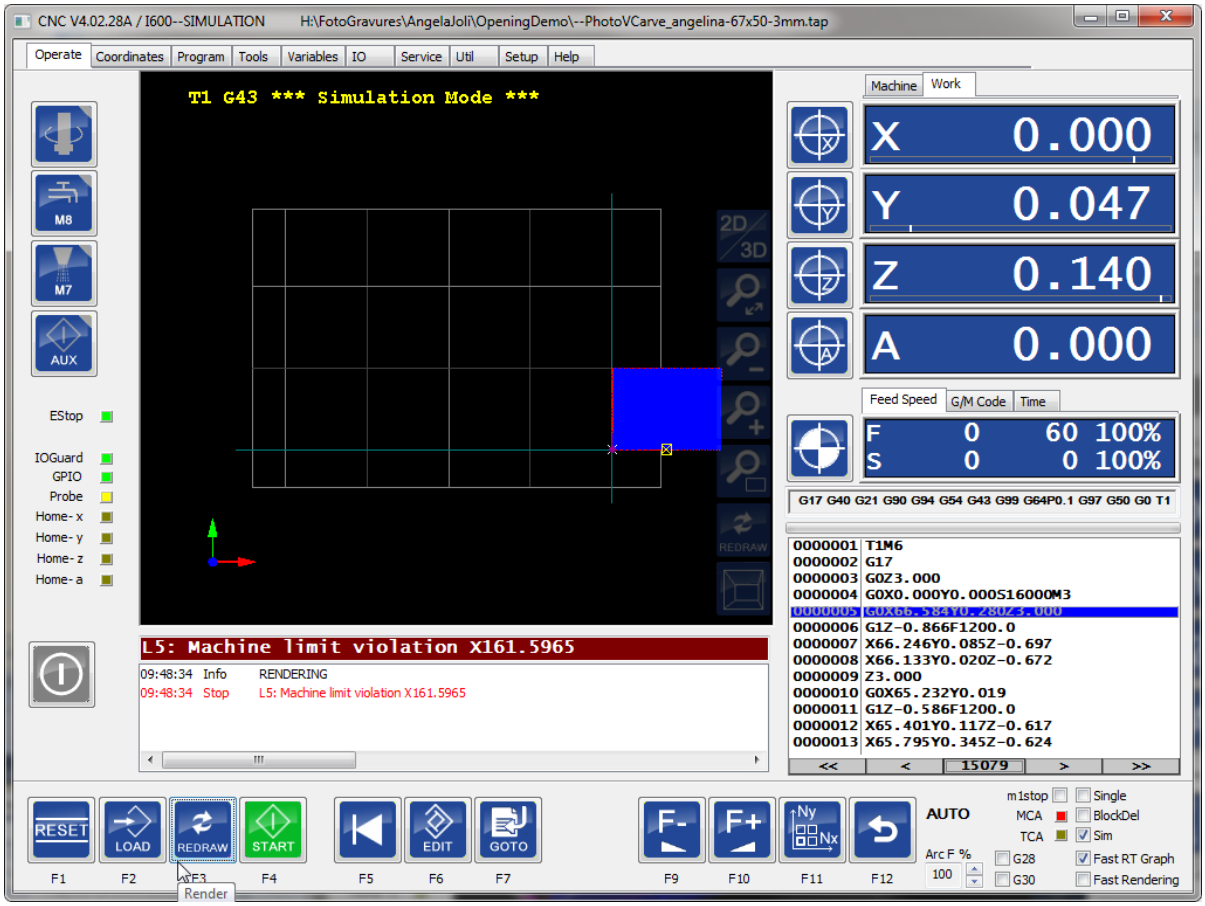
The file is fully parsed through the g-code interpreter and the tool path is shown in the graphic window:



Using the mouse (ctrl+left mouse), you can rotate the tool-path and see it 3D.
Using the left mouse, you can PAN. Using the right mouse you can ZOOM:
It can be that while loading you get a collision error, this means that the tool-path does not fit on the machine, because the work zero point is not at a correct position. he Cian/Light blue colored line's indicate the Work-Zero point. Jog to the left and set zero X further to the left using the button besides the X readout is the easiest way to do this.

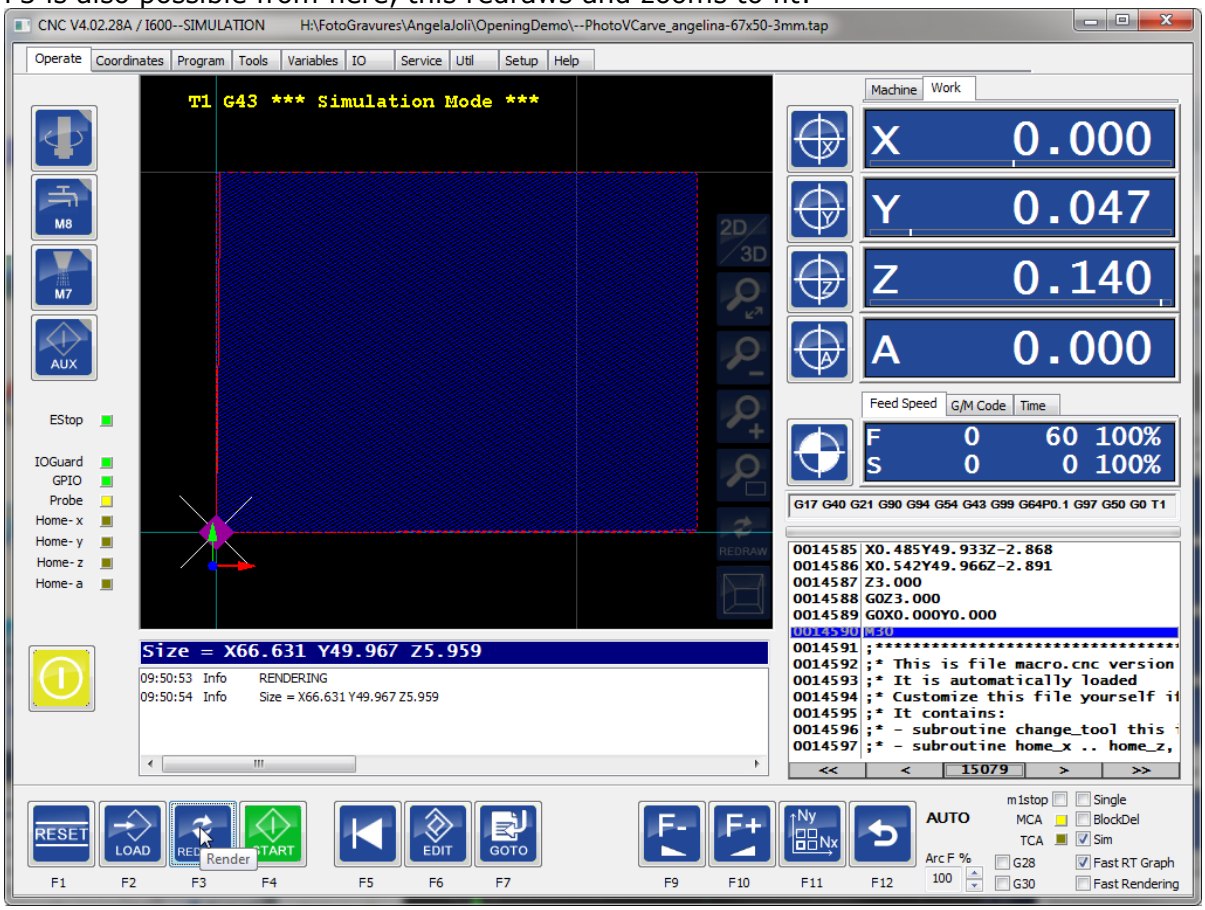


The yellow rectangle shows the first place where the collision is discovered. We see here clearly that a part of the tool-path is outside the machine area, a message is given showing the line number L5 in this case where the collision occurred. The easiest way to shift now is to jog to the place where you want to have the origin, the ctual place of the work-coordinate system origin is shown as the cyan lines for X and Y.



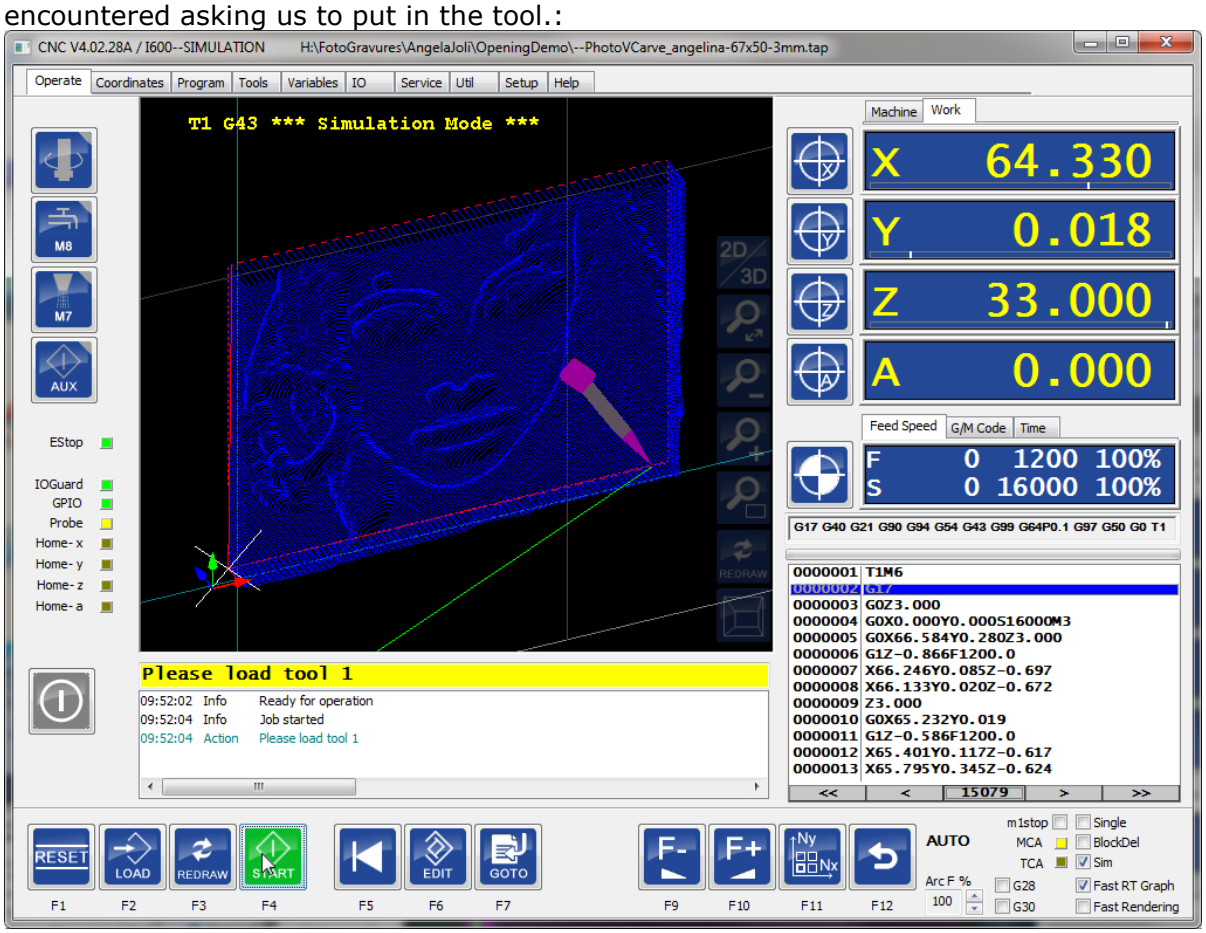
We see that the tool path fits without collision and we see the delta's in X, Y, Z, which is the size of the tool path.

F3 is also possible from here, this redraws and zooms to fit:



Now we can press run (F4) to run the program.

We have no automatic tool changer, so the program stops when a tool change is encountered asking us to put in the tool.:

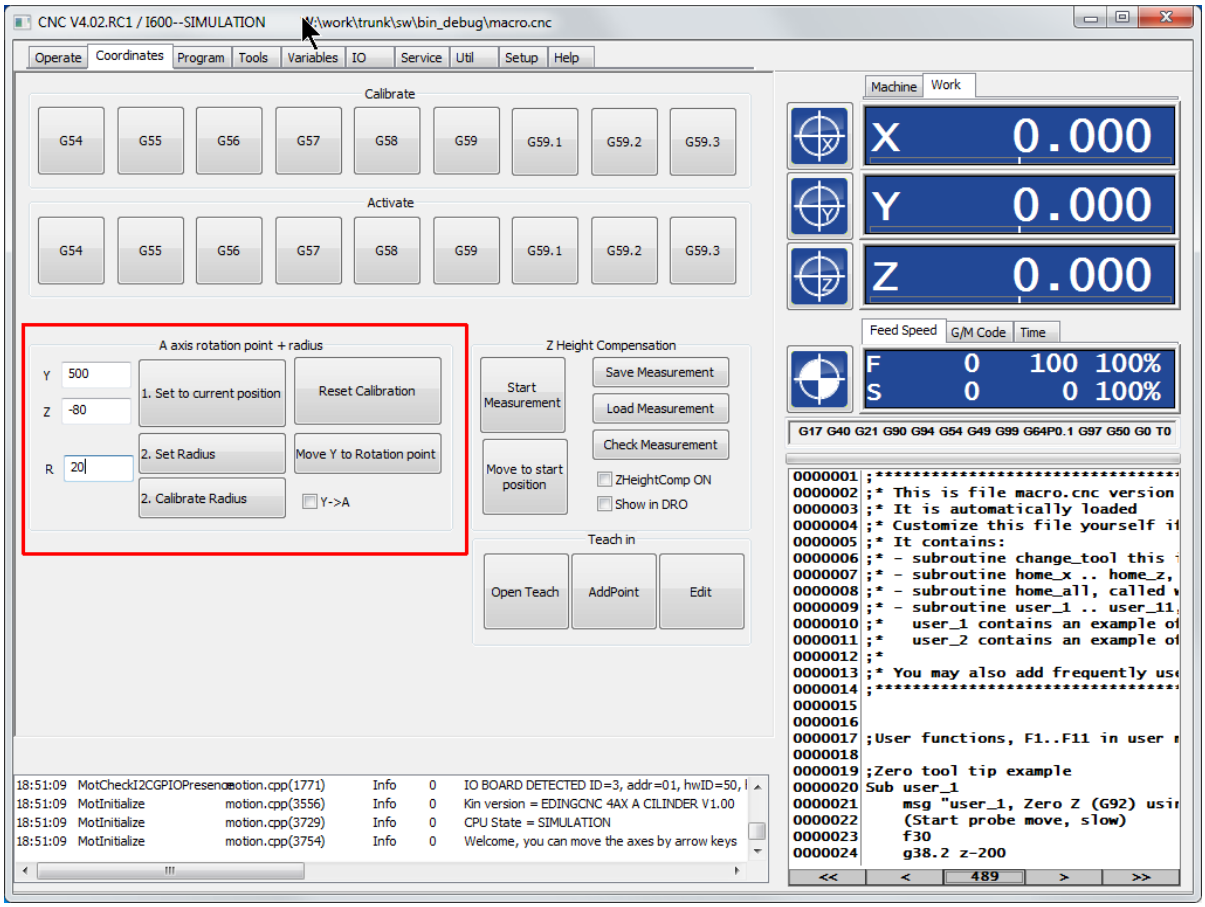


The tool is already in, so we press F4 again, the program will continue and our machine is working.
We see the tool path being drawn real-time on the screen:

If you check the G28 or G30 checkbox, then the machine will return to its G28 or G30 position when the job is done. You can specify those positions in the variables tab.

2.2.5.4 MAPPING X/Y G-CODE TO CYLINDER ON A-AXIS

This is useful if you have a 4th axis and want to mill on the outside of a cylinder as if it was the X-Y plane. You must have setup the A axis as "4THMILL". This is possible with a normal XY G-Code file. EDINGCNC will perform the mapping for you. Some calibration needs to be done to make this work. It is done on the new coordinates page. The location of the A axis rotation point needs to be set and also the radius of the work piece needs to be set first.



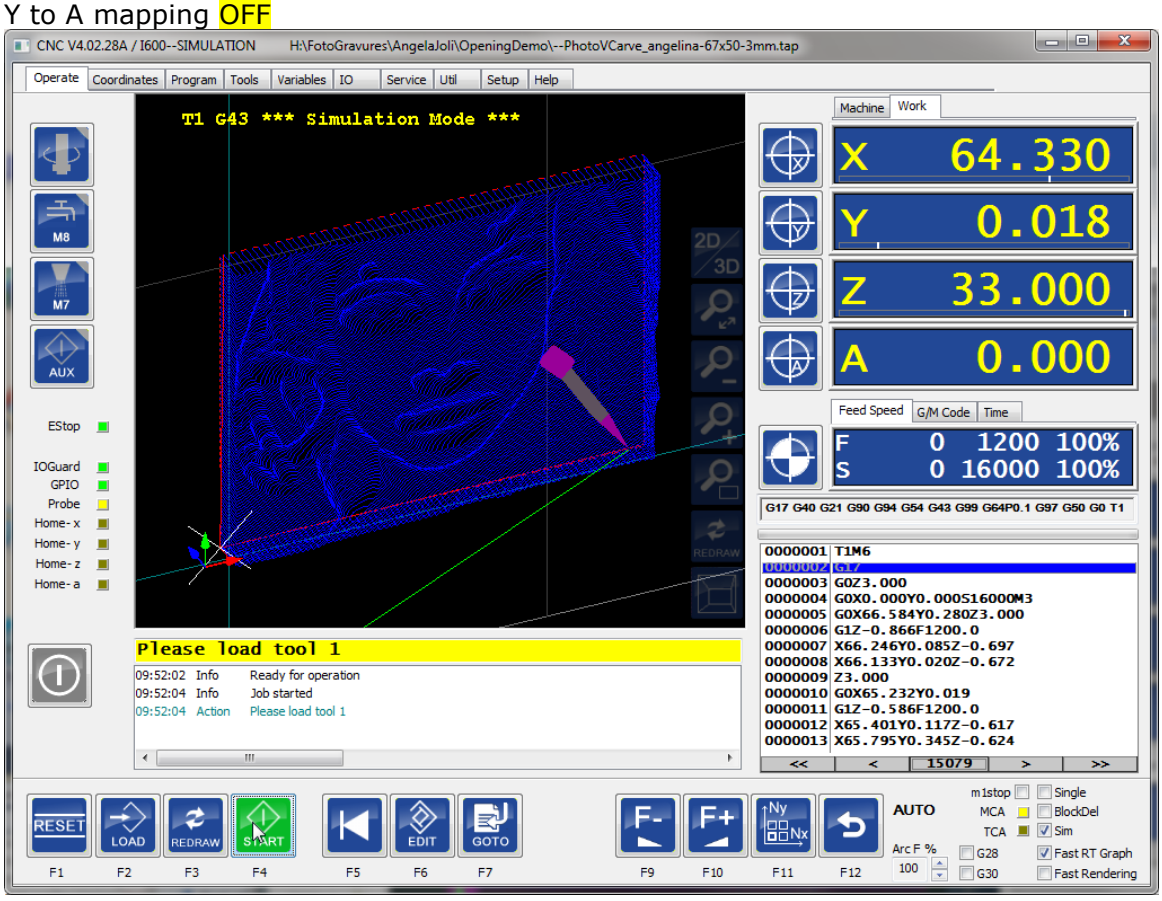
First we set the location of the A rotation point:
We move by jogging or MDI to the center point of rotation of the A axis, only Z and Y are important here. Press button "1. Set to current position". Done, center is set.

Next we set the outside radius of the work piece:
There are 2 possibilities to do this: 1. Just type the radius if you know it and press "Set Radius" Or we second possibility we move the Z up and touch the outside of the material with the tool bit and press "2. Calibrate Radius".

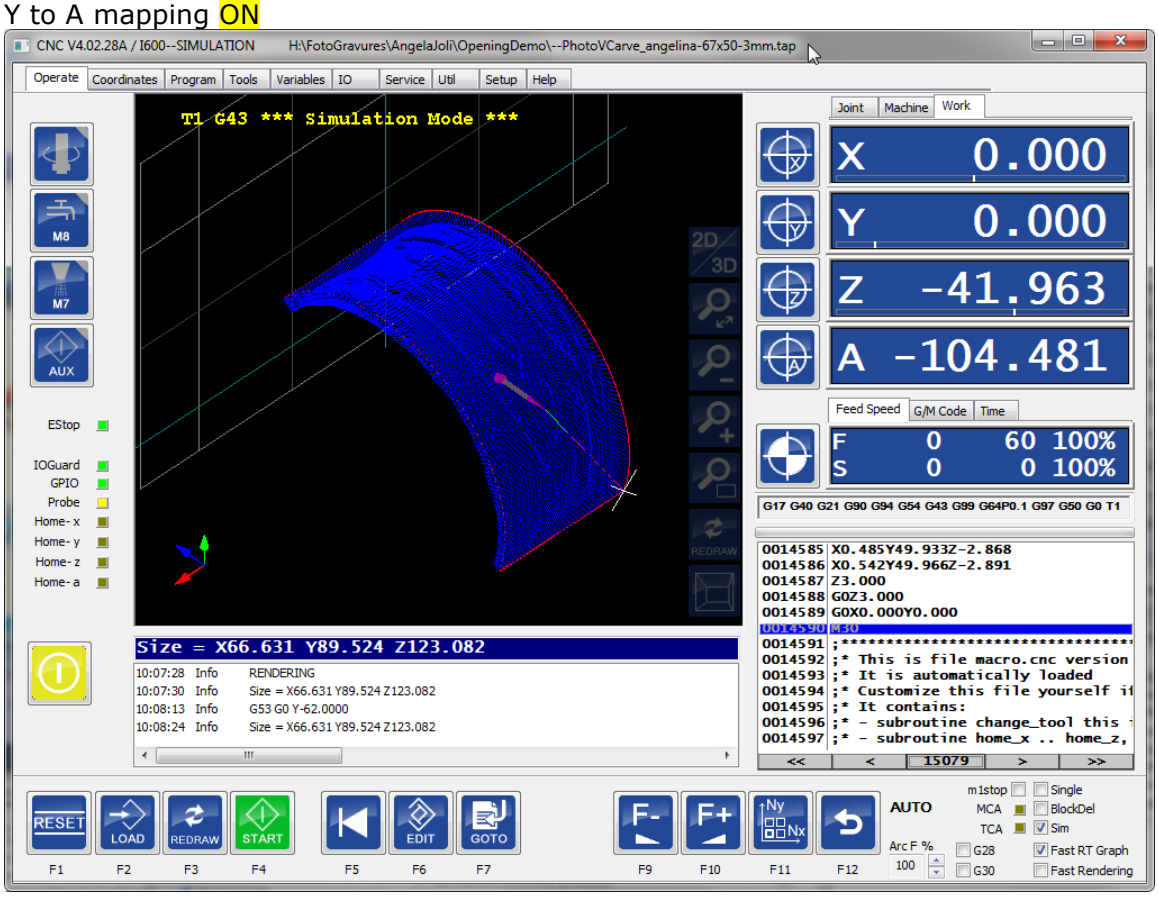
Calibration is done. First we need to be sure that our Y axis is at the correct position before we switch on the Y->A mapping. We need to do this now because when the mapping is ON, Y can no longer be moved, as Y is now mapped to A. You can press "Move Y to rotation point the do the movement.

The mapping can be switched on now. Then we can load a standard G-Code file with XYZ coordinates. Below we see how it looks in the graphic.

Y to A mapping OFF



Y to A mapping ON

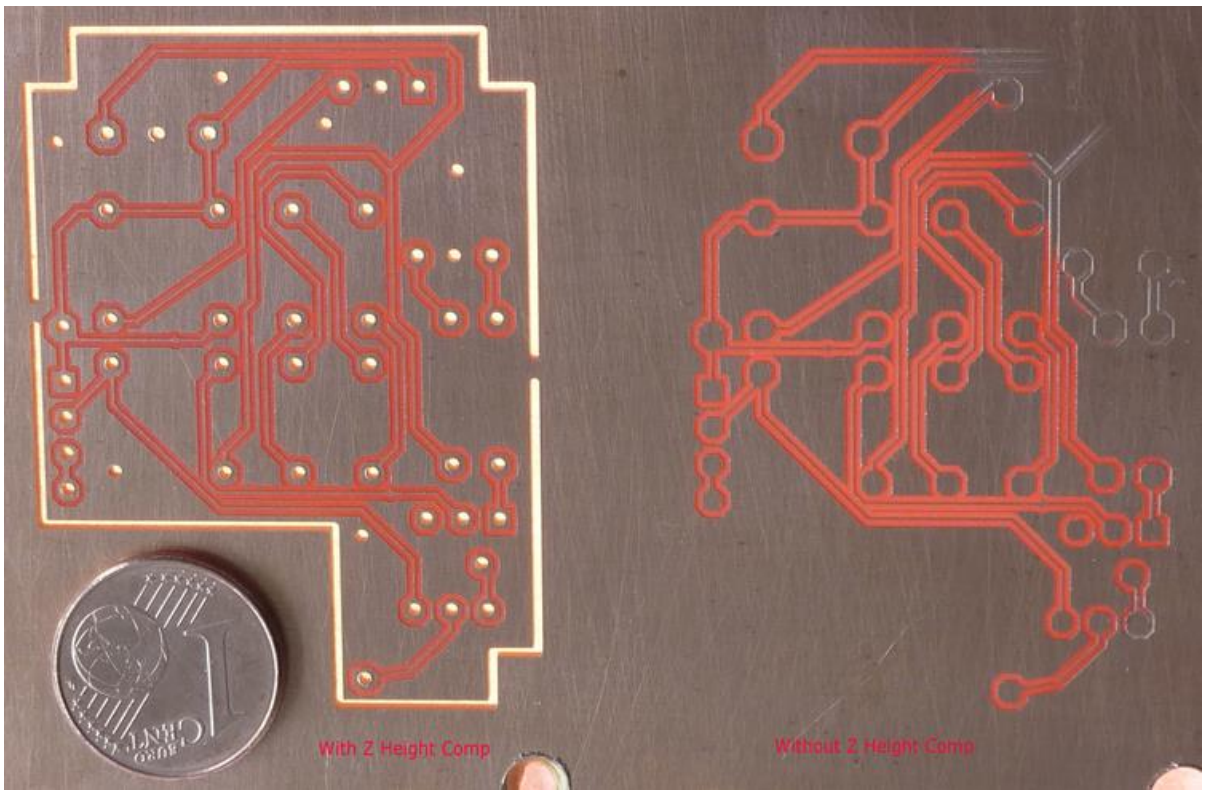


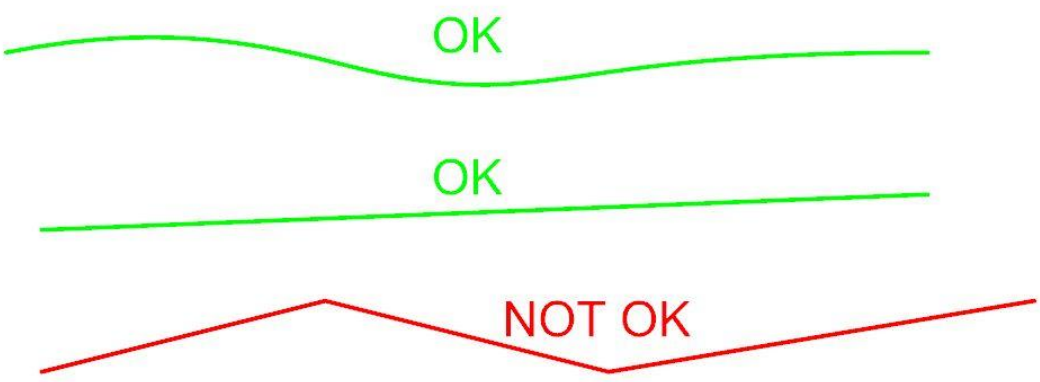
2.2.5.5 MILLING UN-EVEN SURFACES

Intended use

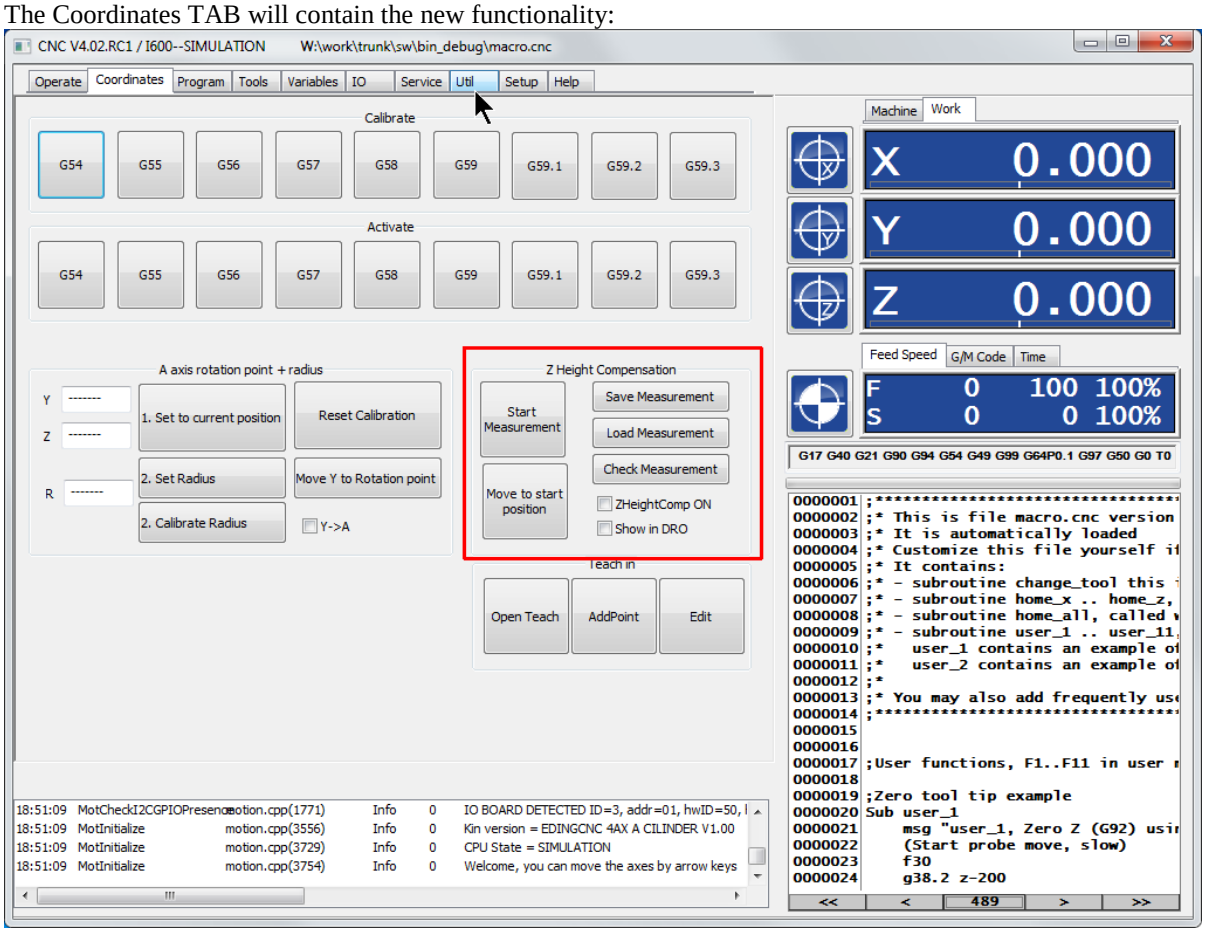
The compensation is intended for relative small compensation with natural smooth behavior.
The compensation profile is directly added to the motion of the Z axis. The acceleration profile therefore is determined by the shape of the compensation.
If the compensation is not continuous, then depending of the quantity of it and the speed in which the moves are done this may lead to position loss with open-loop stepper motor systems or position following error with closed loop systems.

Milling PCB's is one example





The Coordinates TAB will contain the new functionality:


CNC V4.02.RC1 / I600--SIMULATION W:\work\trunk\sw\bin_debug\macro.cnc
Operate Coordinates Program Tools Variables IO Service Util Setup Help
Calibrate
G54 G55 G56 G57 G58 G59 G59.1 G59.2 G59.3
Activate
G54 G55 G56 G57 G58 G59 G59.1 G59.2 G59.3
A axis rotation point + radius
Y
Z
R
1. Set to current position
Reset Calibration
2. Set Radius
Move Y to Rotation point
2. Calibrate Radius
Y->A
Z Height Compensation
Start Measurement
Save Measurement
Load Measurement
Check Measurement
Move to start position
ZHeightComp ON
Show in DRO
Teach in
Open Teach
AddPoint
Edit
Machine Work
X 0.000
Y 0.000
Z 0.000
Feed Speed G/M Code Time
F 0 100 100%
S 0 0 100%
G17 G40 G21 G90 G94 G54 G49 G99 G64P0.1 G97 G50 G0 T0
0000001 ;*********
0000002 ;* This is file macro.cnc version
0000003 ;* It is automatically loaded
0000004 ;* Customize this file yourself if
0000005 ;* It contains:
0000006 ;* - subroutine change_tool this
0000007 ;* - subroutine home_x .. home_z,
0000008 ;* - subroutine home_all, called
0000009 ;* - subroutine user_1 .. user_11,
0000010 ;* user_1 contains an example of
0000011 ;* user_2 contains an example of
0000012 ;*
0000013 ;* You may also add frequently us
0000014 ;*********
0000015
0000016
0000017 ;User functions, F1..F11 in user
0000018
0000019 ;Zero tool tip example
0000020 Sub user_1
0000021 msg "user_1, Zero Z (G92) usi
0000022 (Start probe move, slow)
0000023 f30
0000024 g38.2 z-200
<< < 489 > >>
18:51:09 MotCheckI2CGPIOPresence motion.cpp(1771) Info 0 IO BOARD DETECTED ID=3, addr=01, hwID=50,
18:51:09 MotInitialize motion.cpp(3556) Info 0 Kin version = EDINGCNC 4AX A CILINDER V1.00
18:51:09 MotInitialize motion.cpp(3729) Info 0 CPU State = SIMULATION
18:51:09 MotInitialize motion.cpp(3754) Info 0 Welcome, you can move the axes by arrow keys

“Start measurement”
will popup an interpreter dialog for the automatics measurement using a toch probe. It is explaned on next page

“Move to start position”
will move X and Y to the first measured position, there where the compensation is zero.

“Save measurement”
Will open a file-save dialog and allows to save the mesurement data.

“Load measurement”
Shows a file-open dialog to load existing measurement data.

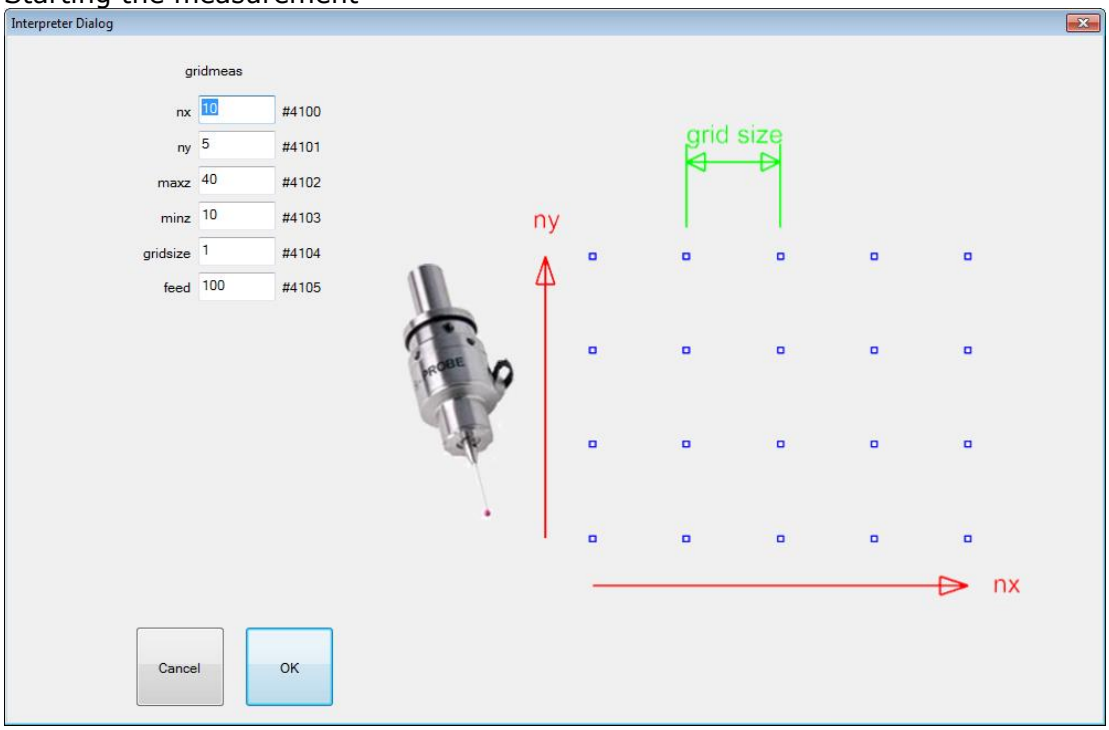
“Check measurement”
Wil show some statics of the measured data, the max and minimum correction values and at which place they are.

“ZheightComp ON”
Wil switch the compensation ON/OFF.

It will be shown in the position read out, the compensation value is shown above the normal work position of Z.

Now you can do normal XY engraving and while the Z is compensated using the measurement values.

Starting the measurement



You specify the number of measurement in X and Y.
The height to which the probe should move to when going to the next point.
The minimum Z value to which the probe should move to during the measurement (G38.2).
The size of the grid, distance between the measuring points.
And the feed for the down movement during G38.2. All other moves use G0 for highest speed.

The measurement starts at WORK X0 Y0. maxZ and Min Z are also WORK coordinates.
The measurement data itself is in motor coordinates, so after the measurement you can freely zero anywhere else on the work piece.

This is the subroutine to be added in the standard macro.cnc for this purpose.
If you have a self-modified macro.cnc, you can copy from default_macro.cnc to your own macro.cnc

```
sub zhcmgrid
;;;;;;;;;;;;;
;probe scanning routine for eneven surface milling
;scanning starts at x=0, y=0

  if [#4100 == 0]
   #4100 = 10  ;nx
   #4101 = 5   ;ny
   #4102 = 40  ;max z
   #4103 = 10  ;min z
   #4104 = 1.0 ;step size
   #4105 = 100 ;probing feed
  endif

  #110 = 0    ;Actual nx
  #111 = 0    ;Actual ny
  #112 = 0    ;Missed measurements counter
  #113 = 0    ;Number of points added
  #114 = 1    ;0: odd x row, 1: even xrow

  ;Dialog
  dlgmsg "gridMeas" "nx" 4100 "ny" 4101 "maxZ" 4102 "minZ" 4103 "gridSize" 4104 "Feed" 4105
```

```
  if [#5398 == 1] ; user pressed OK
    ;Move to startpoint
    g0 z[#4102];to upper Z
    g0 x0 y0 ;to start point

    ;ZHCINIT gridSize nx ny
    ZHCINIT [#4104] [#4100] [#4101]

    #111 = 0    ;Actual ny value
    while [#111 < #4101]
        if [#114 == 1]
          ;even x row, go from 0 to nx
          #110 = 0 ;start nx
          while [#110 < #4100]
            ;Go up, goto xy, measure
            g0 z[#4102];to upper Z
            g0 x[#110 * #4104] y[#111 * #4104] ;to new scan point
            g38.2 F[#4105] z[#4103];probe down until touch

            ;Add point to internal table if probe has touched
            if [#5067 == 1]
              ZHCADDPOINT
              msg "nx="[#110 +1]" ny="[#111+1]" added"
              #113 = [#113+1]
            else
              ;ZHCADDPOINT
              msg "nx="[#110 +1]" ny="[#111+1]" not added"
              #112 = [#112+1]
            endif

            #110 = [#110 + 1] ;next nx
          endwhile
          #114=0
        else
          ;odd x row, go from nx to 0
          #110 = [#4100 - 1] ;start nx
          while [#110 > -1]
            ;Go up, goto xy, measure
            g0 z[#4102];to upper Z
            g0 x[#110 * #4104] y[#111 * #4104] ;to new scan point
            g38.2 F[#4105] z[#4103];probe down until touch

            ;Add point to internal table if probe has touched
            if [#5067 == 1]
              ZHCADDPOINT
              msg "nx="[#110 +1]" ny="[#111+1]" added"
              #113 = [#113+1]
            else
              ;ZHCADDPOINT
              msg "nx="[#110 +1]" ny="[#111+1]" not added"
              #112 = [#112+1]
            endif

            #110 = [#110 - 1] ;next nx
          endwhile
          #114=1
        endif

      #111 = [#111 + 1] ;next ny
    endwhile

    g0 z[#4102];to upper Z
    ;Save measured table
    ZHCS zHeightCompTable.txt
    msg "Done, "#113" points added, "#112" not added"

  else
    ;user pressed cancel in dialog
    msg "Operation canceled"
  endif
endsub
```

Milling un-even cylinders with y->a mapping on also works.
In that case the measurement must be done using with y->a mapping on.

The compensation works only in the measured range. So of the mapping is measured between 0-360 degrees for A, it will not compensate for e.g. 370 degrees.

The Z height compensation interpreter commands

For your own use if you want to customize the working:

ZHCINIT <grid size in mm> <number of points in X> <number of points in Y>
This is the first command required before starting a measurement, it will reserve the correct amount of memory to store the measured points.

Using G38.2 will do the actual measurement.

ZHCADDPOINT will add the last measured point to the data.

ZHCS <fileName> will store the data to a file.

ZHCL <fileName> will load the data from a file.

ZHC [ON | OFF] will switch the compensation on.

#5051 contains 1 if ZHC is ON and 0 if OFF. This can be used for run time checks if the compensation is on. In case of tool change, you will probably want to switch it off and switch it back ON after the tool change.

2.3 LINEAR PITCH COMPENSATION

This is a way to improve the accuracy of the machine when the linear displacements are not exactly correct. E.g. cheap rolled ball bearing spindles may have an inaccuracy of several 0.1 mm at a meter length. Also the pitch may vary a bit depending on the position. This compensation feature allows to correct this.

The compensation can be switched on by manually editing the cnc.ini file (contains all settings).
Under each joint settings [JOINT_0] is the first, usually your X axis you find 2 settings:

- **pitchCompensationOn** = 1 (1 to switch compensation ON, 0 to switch OFF)
- **pitchCompensationFileName** = "Joint-X-pitchCompTable.txt" (name of the file with the correction table)

when you switch on the compensation and the compensation table does not exist, one is created as example for you, it is only an example to show the syntax. You need to adopt it for your machine.

This is an example of a correction table:

```
;Pitch correction table for axis X
;This table contains 6 correction points
;machine-position calibrated-Position
   0.0000    0.0000
  50.0000   50.01000
 200.0000  200.02000
 300.0000  300.03000
 400.0000  400.04000
 500.0000  500.05000
```

The left value is the position of the machine.
The right value is the calibrated position that you have obtained by measuring it.
The table must be sorted, small to big values. Note that 0 is bigger than -100, so if you have negative numbers the highest negative number comes first.

You can make the compensation value visible by checking Show in DRO on the coordinates window. You will see this in the DRO:

The small number above the position shows the actual compensation value.

2.4 SPEED-PWM COMPENSATION

The speed for a spindle is controlled by the PWM output.
The PWM is converted to an analogue signal which is fed to the VFD (Variable frequency drive). There are often non linearity's involved.
This cause that the programmed speed and the real speed does not match correctly.
This software feature allows to compensate this.

The compensation can be switched on by manually editing the cnc.ini file (contains all settings).
Under each spindle settings [SPINDLE_0] is the first, usually your main spindle (M90) axis you find 2 settings:

- **pwmCompensationOn**= 1 (1 to switch compensation ON, 0 to switch OFF)
- **pwmCompensationFileName**= " Spindle-0-pwmCompTable.txt" (name of the file with the correction table)

when you switch on the compensation and the compensation table does not exist, one is created as example for you, it is only an example to show the syntax. You need to adopt it for your machine.

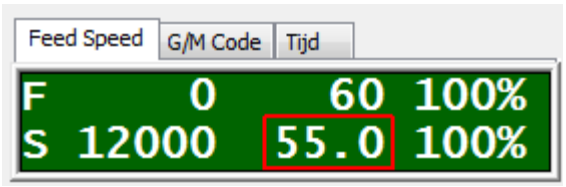
This is an example of a correction table:

```
;Speed PWM calibration table for spindle 0
;This table contains 15 correction points
;Speed    PWM Percentage
    0.00      0.00
  720.00      8.00
 1146.00     10.00
 3468.00     20.00
 5832.00     30.00
 6000.00     31.00
 8280.00     40.00
10740.00     50.00
13260.00     60.00
15780.00     70.00
18000.00     79.00
18120.00     80.00
21060.00     90.00
24000.00    100.00
```

The number on the left side is the Spindle Speed and on the right side the required PWM value 0-100%.
You can see the PWM value in the IO screen and you can make it also visible at the main screen instead of the programmed speed.

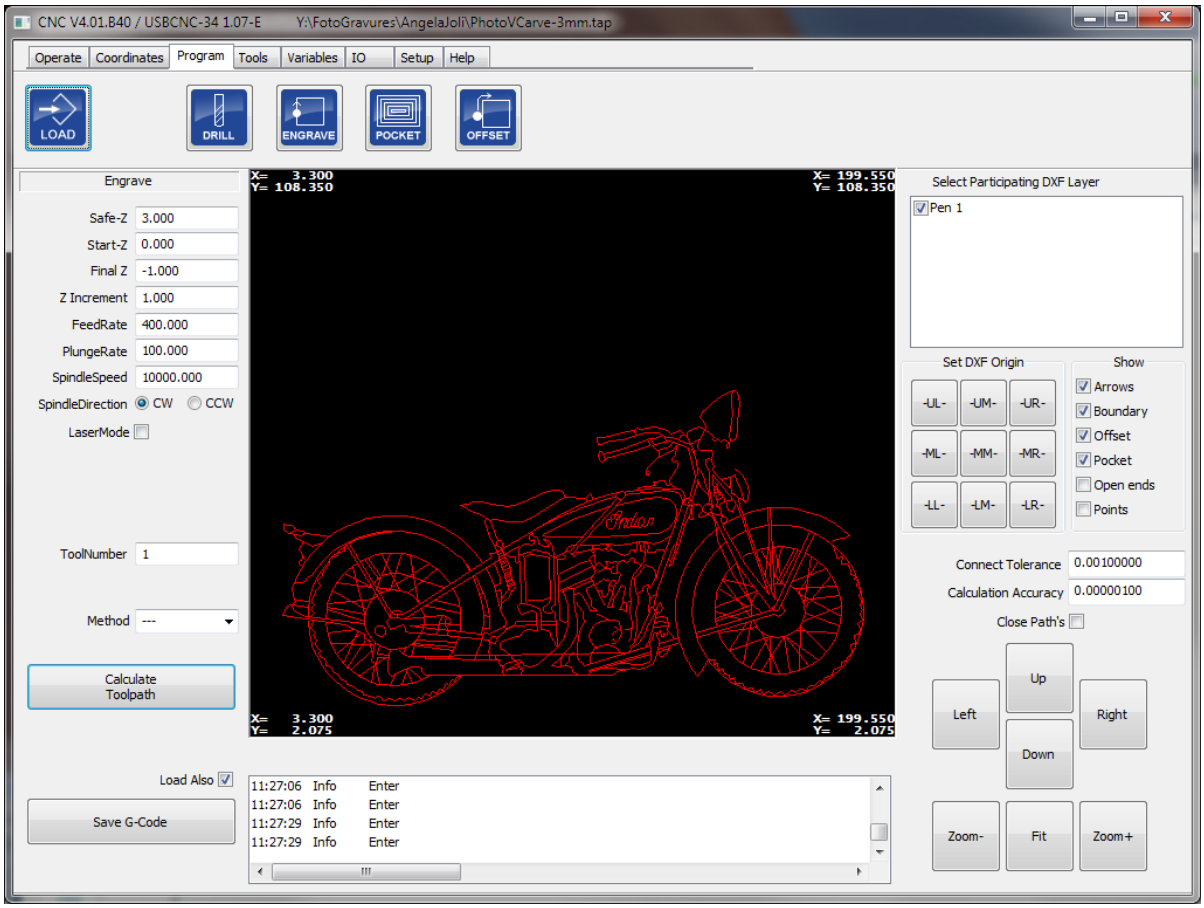
The setting is under [USERINTERFACE]
;0=programmed speed, 1=PWM value, 2=analogIn1, 3=analogIn2, 4=analogIn3

- **showInProgSpeed** = 1
- **showInProgSpeedAnaMulFactor** = 1.0000



As you can see you can also choose to show an analogue input with a multiplication factor, this can be used when the spindle has a power used signal.

2.5 PROGRAM PAGE, DXF AND HPGL IMPORT



EDINGCNC uses a build in CAD/CAM library for these advanced import functions. You can load a file and then perform one of these operations:

	Loads a DXF or HPGL file
	Select engraving, this is milling over the lines from the drawing.
	Drilling, draw points in the DXF file to use this.
	Select profiling, this is for milling out objects and taking the tool diameter into account. (*)
	This is for pocketing, to mill out the complete object. (*)

(*) these options may not be available.

After loading a DXF file, all layers will be visible. You can unselect layers at the right side, such that you see only the part that you want to use. You also can change the origin of the drawing by pressing the appropriate

button under the layer selection list box. The positions of the buttons give the positions of the origin. So e.g. when you press the upper right button, then the most upper right position of the drawing will become x=0, y=0 when milling.

The DXF import supports:

- Lines
- Arcs
- Circles
- Poly lines with arcs
- Points for drilling

The workflow of using these features is:

1) Load drawing
2) Select the correct layers
3) Apply origin offset if wanted
4) Set correct parameters
5) Calculate tool path
6) Save tool path and optionally immediately load it for milling.

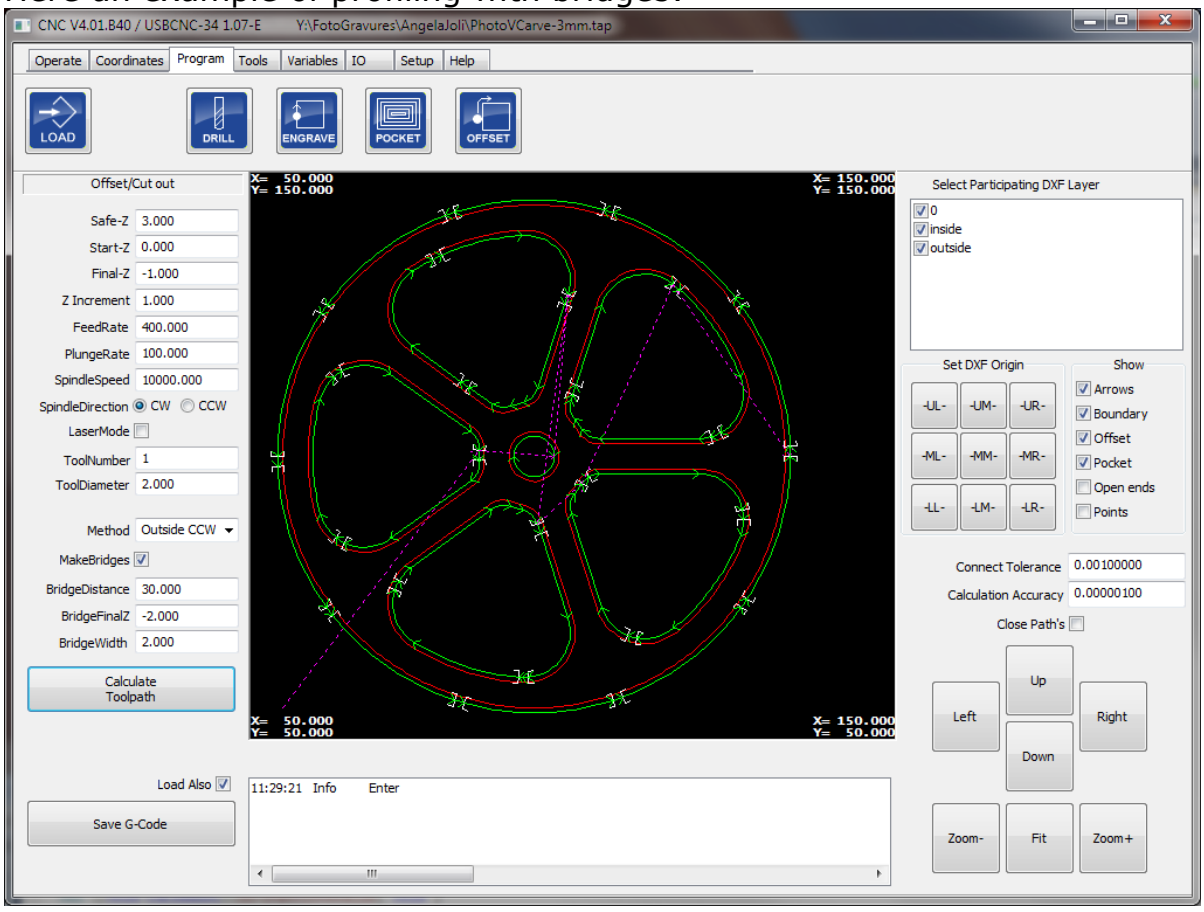
Parameters involved:

Save-Z	When moving from one region to another, the machine goes to this height.
Start-Z	Z value where the tool touches the material to be machined.
Final-Z	Z value specifying the milling depth lowest Z value. Final Z must be lower than Start Z.
Z Increment	This specifies the step size when machining in passes.
Feed rate	Milling feed (F) in mm/min
Plunge rate	Feed (F) that the Z moves down into the material also mm/min
Spindle speed	S value for spindle.
CW/CCW	Spindle direction (M3/M4)
Tool number	This is only used for the M6 tool change command
Tool Diameter	Diameter of the tool for the offset and pocketing calculations.
Method	Outside/inside/clockwise/counterclockwise operation
Finish allowance.	Material that is left for the finishing pass when pocketing. This finishing pass is at full depth for getting a clean edge.
Step size	Step oversize for pocketing, this value should be lower

	than the tool diameter.
Laser mode	For profiling, when switched on, the tool will be switched off when moving from one region to another.
Make bridges	Leave small pieces of material, that prevent you object from falling out (and get damaged) when profiling.
Bridge distance	Approx distance, the exact distance is calculated such that all bridges have equal distance.
BridgeFinal Z	Lowest Z value for bridge, this value should be between startZ and finalZ
BridgeWidth	The width of a bridge.

When the parameters are set, press calculate tool path, it will be visualized on the screen.

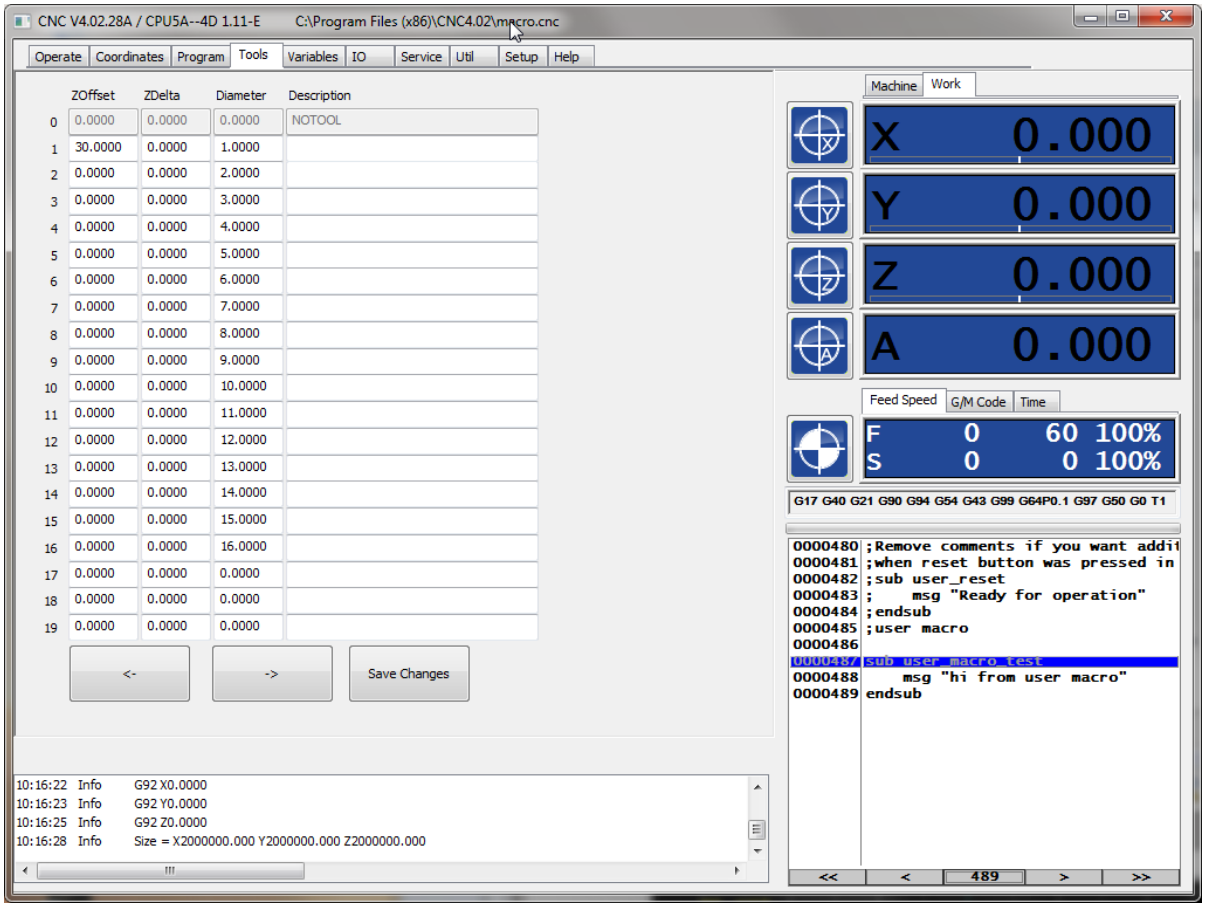
Here an example of profiling with bridges:



Note: The offset and pocket calculation might not always work, this is usually because of small errors in the drawing like lines over each other or not connecting lines. Experimenting with the Calculation Accuracy might help. Also check/correction of your drawing may help.
The engraving function is robust and will always work.

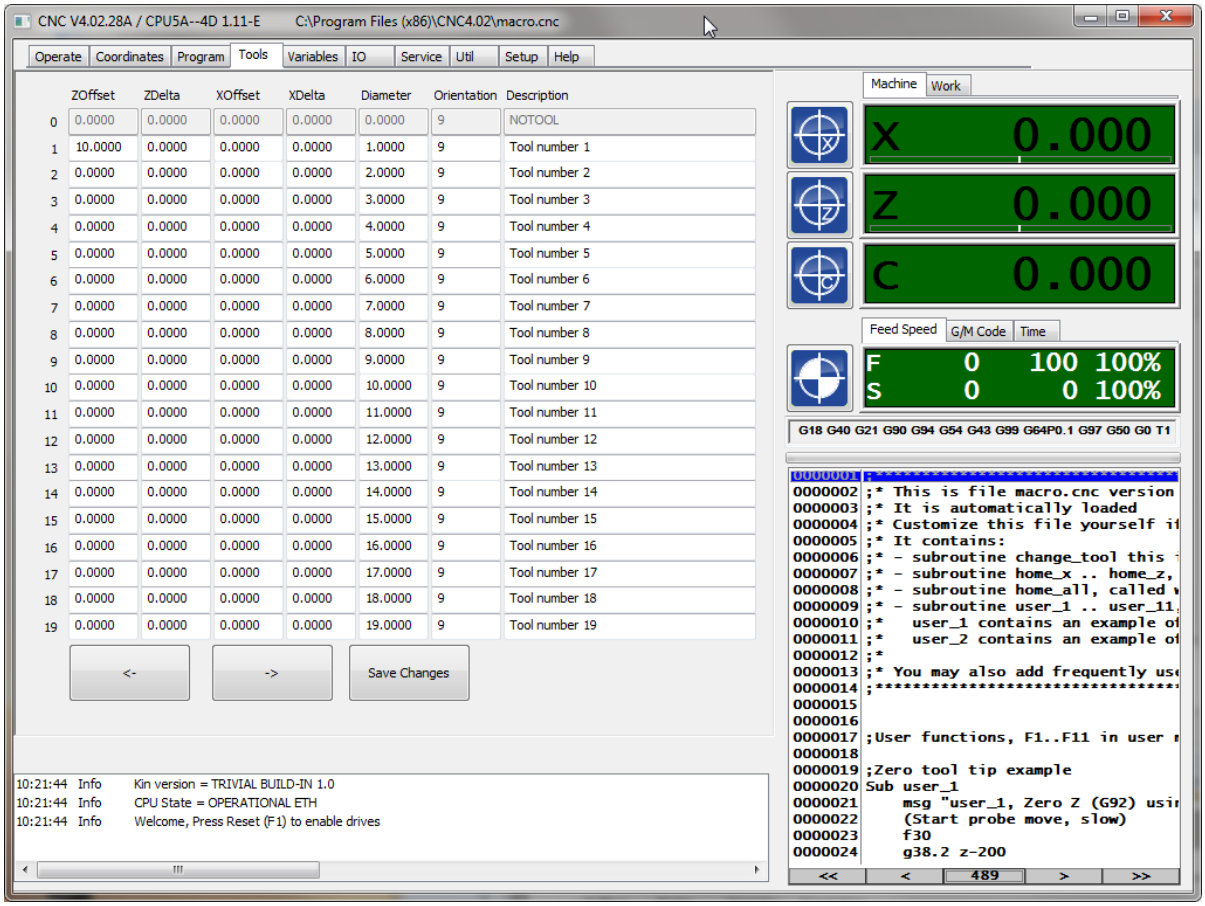
2.6 TOOLS PAGE

2.6.1 Milling



In this view you can define 99 tools with a Z-Offset(length) ZDelta (Delta due to Wear in Z length), diameter and description. The tool information is used when you use the tool radius and or tool length compensation functions of the G-Code interpreter commands (G40 – G43). See chapter 3.6. and further.
ZDelta is new and can be used e.g. if you see that you haven't milled deep enough, simple set a negative value to ZDelta and you run the program again. G43 must be active to use this.

2.6.2 **Turning**



As you can see, there are additional parameters for turning, X-Offset X-Delta and Orientation.

2.6.3 **Tool change**

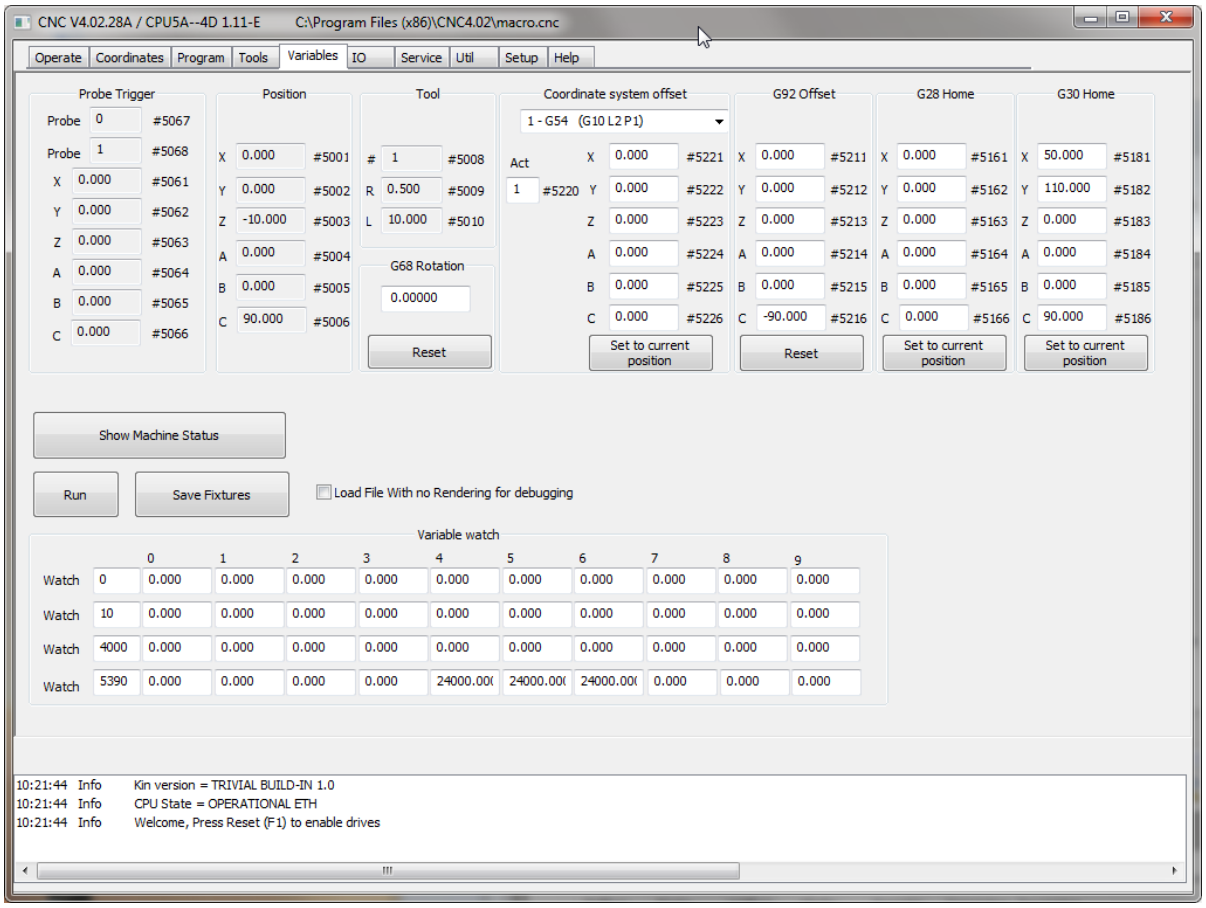
A tool change is performed in G-Code by M6 Tx where Tx is the new tool number. Tool number 0 means no tool.
Normally the program is stopped on a tool change, with a user message to change the tool, pressing run again will continue the program. If you don’t want the program to stop, check AutoToolChange in the automatic menu bar. This setting is saved when you press save INI file in the setup screen.

2.6.4 **Automatic user defined Tool change ATC**

When you want to define you own tool change cycle, you can edit the file “macro.cnc” in the EDINGCNC directory. When an M6 Tx is encountered, this is translated to a GOSUB of subroutine change_tool in the “macro.cnc” file. This subroutine then calls further subroutines drop_tool_x and pick_tool_x, if you have a tool changer, you can add extra movements to the right tool position and control I/O for actually changing the tool.

2.6.5 **The variable Page**

This page shows the standard variables used by the G-Code interpreter. It also contains 4 watches to show your own variables if you are going to use the extended programming features. You will understand the meaning of this window after reading the G-Code interpreter functions and extended programming with variables.

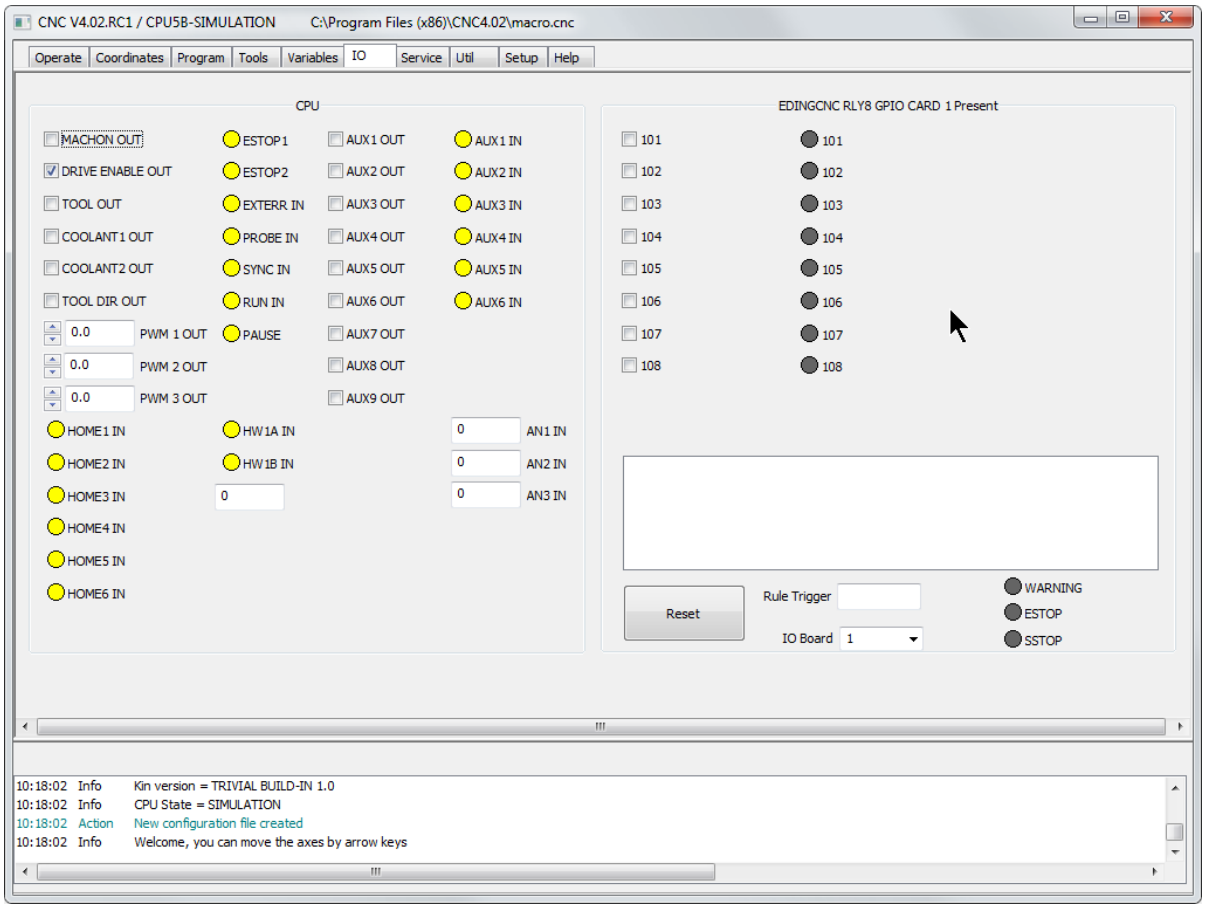

CNC V4.02.28A / CPU5A--4D 1.11-E C:\Program Files (x86)\CNC4.02\macro.cnc
Operate Coordinates Program Tools Variables IO Service Util Setup Help
Probe Trigger
Position
Tool
Coordinate system offset
1 - G54 (G10 L2 P1)
G92 Offset
G28 Home
G30 Home
G68 Rotation
Reset
Set to current position
Show Machine Status
Run
Save Fixtures
Load File With no Rendering for debugging
Variable watch
10:21:44 Info Kin version = TRIVIAL BUILD-IN 1.0
10:21:44 Info CPU State = OPERATIONAL ETH
10:21:44 Info Welcome, Press Reset (F1) to enable drives

The G68 rotation can be reset with the reset button under G68 Rotation.
This is the same as entering G69 in MDI.

The G54 .. G59.3 offsets can be set by entering values and pressing enter.
The G54.. G59.3 X,Y values can be defined as zero at current machine position by pressing the button. This works similar to G92 offset.
The MDI equivalent for setting G55 offsets is G10 L20 P2 X0 Y0.

G92 is normally used for zeroing the machine at work piece coordinates. you can reset all offsets here to zero. The G28 and G30 positions can be defined at current location by pressing the associated button.

2.7 IO PAGE



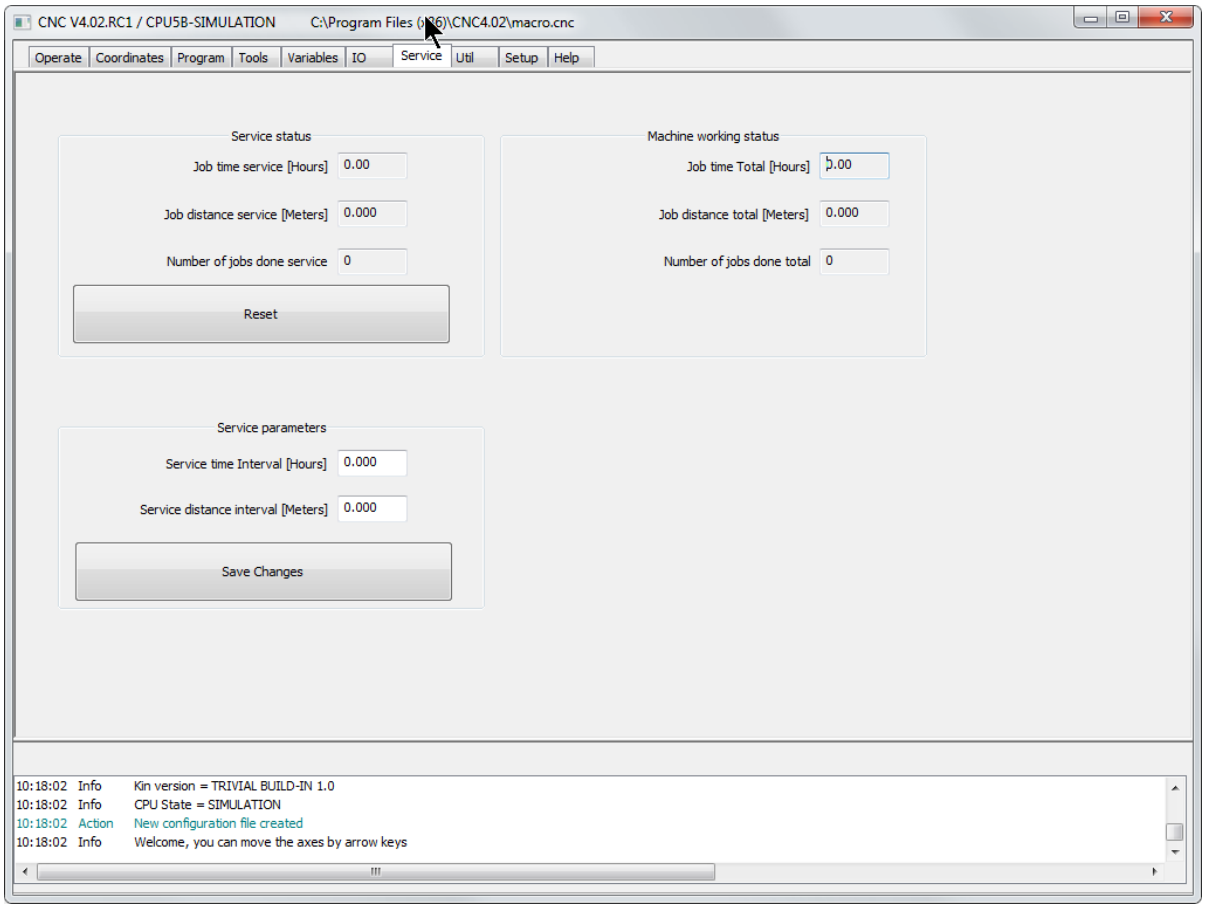
At this page you can monitor and set the I/O signals for CPU and IO extension board if attached. You can see that the GPIO extension board signals can be given a more meaningful name to your application.

The internal GPIO (AUX) can be controlled by M54, M55, M56, e.g. M54 P1 switches output AUX1 on and M55 P1 switches output AUX1 off.

The external GPIO can be controlled also by M54, M55, M56, e.g. M54 P101 switches output 1 of card 1 on.

These M-Codes are further explained in the M Codes chapter of this document.

2.8 SERVICE PAGE



At this page shows how much your machine is operated and if it needs service.
You can see:

Service status

Job time service : It shows the number of hours the machine has performed jobs.
Job distance: The distance the machine has milled in meters.
Number of jobs done service: The number of jobs done.
These values can be reset to zero when the machine gets service with the reset button.

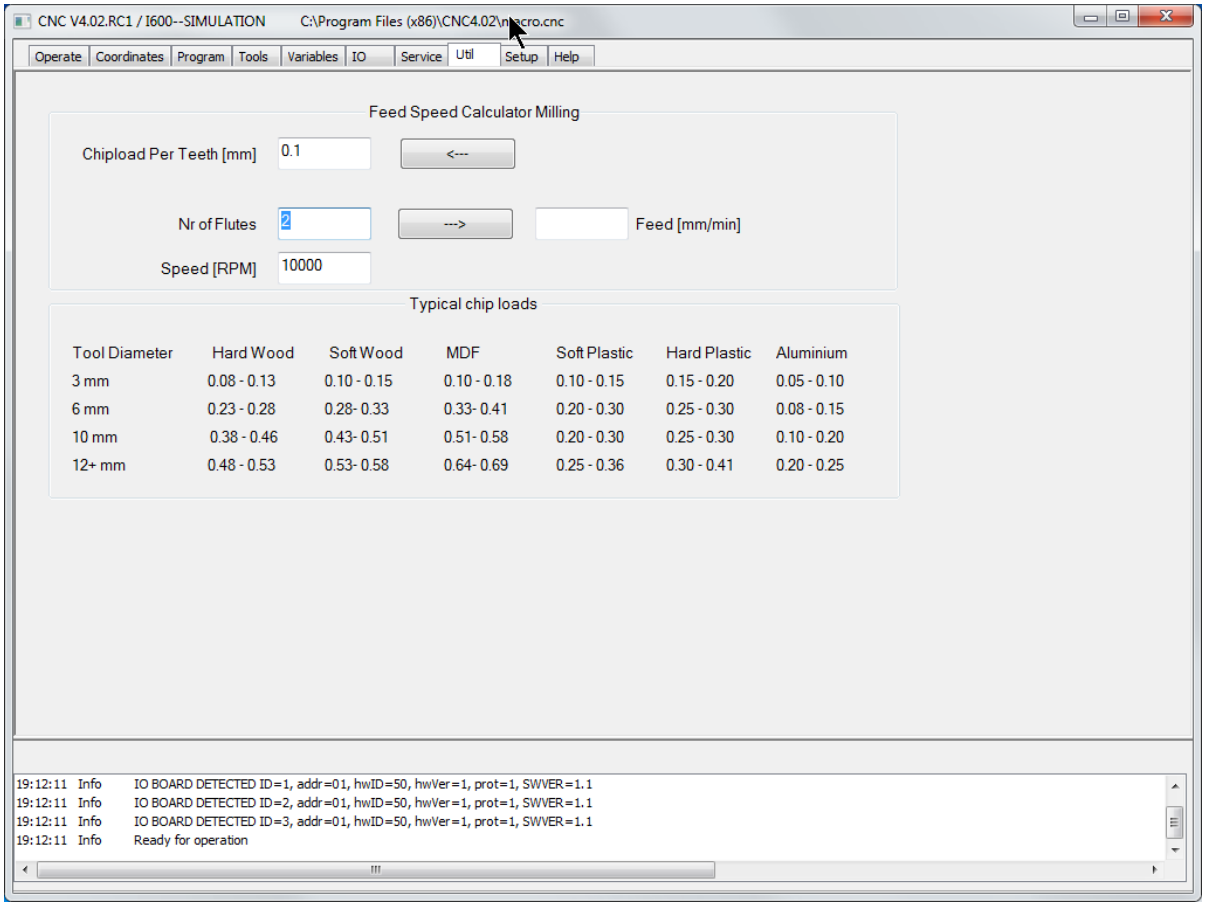
Machine working status

Job time total : It shows the number of hours the machine has performed jobs.
Job distance total: The distance the machine has milled in meters.
Number of jobs done total: The number of jobs done.
These values cannot be reset, it shows the total usage of the machine.

Service parameters

Service Time interval: The time interval for the service, the software will give a message when this is passed at the end of a job.
Service distance interval: The traveled distance for service. Also here the software will give a message to indicate the machine needs service.

2.9 UTIL PAGE, CHIPLOAD AND FEED/SPEED


CNC V4.02.RC1 / I600--SIMULATION C:\Program Files (x86)\CNC4.02\macro.cnc
Operate
Coordinates
Program
Tools
Variables
IO
Service
Util
Setup
Help
Feed Speed Calculator Milling
Chipload Per Teeth [mm]
0.1
<---
Nr of Flutes
2
--->
Feed [mm/min]
Speed [RPM]
10000
Typical chip loads
Tool Diameter Hard Wood Soft Wood MDF Soft Plastic Hard Plastic Aluminium
3 mm 0.08 - 0.13 0.10 - 0.15 0.10 - 0.18 0.10 - 0.15 0.15 - 0.20 0.05 - 0.10
6 mm 0.23 - 0.28 0.28- 0.33 0.33- 0.41 0.20 - 0.30 0.25 - 0.30 0.08 - 0.15
10 mm 0.38 - 0.46 0.43- 0.51 0.51- 0.58 0.20 - 0.30 0.25 - 0.30 0.10 - 0.20
12+ mm 0.48 - 0.53 0.53- 0.58 0.64- 0.69 0.25 - 0.36 0.30 - 0.41 0.20 - 0.25
19:12:11 Info IO BOARD DETECTED ID=1, addr=01, hwID=50, hwVer=1, prot=1, SWVER=1.1
19:12:11 Info IO BOARD DETECTED ID=2, addr=01, hwID=50, hwVer=1, prot=1, SWVER=1.1
19:12:11 Info IO BOARD DETECTED ID=3, addr=01, hwID=50, hwVer=1, prot=1, SWVER=1.1
19:12:11 Info Ready for operation

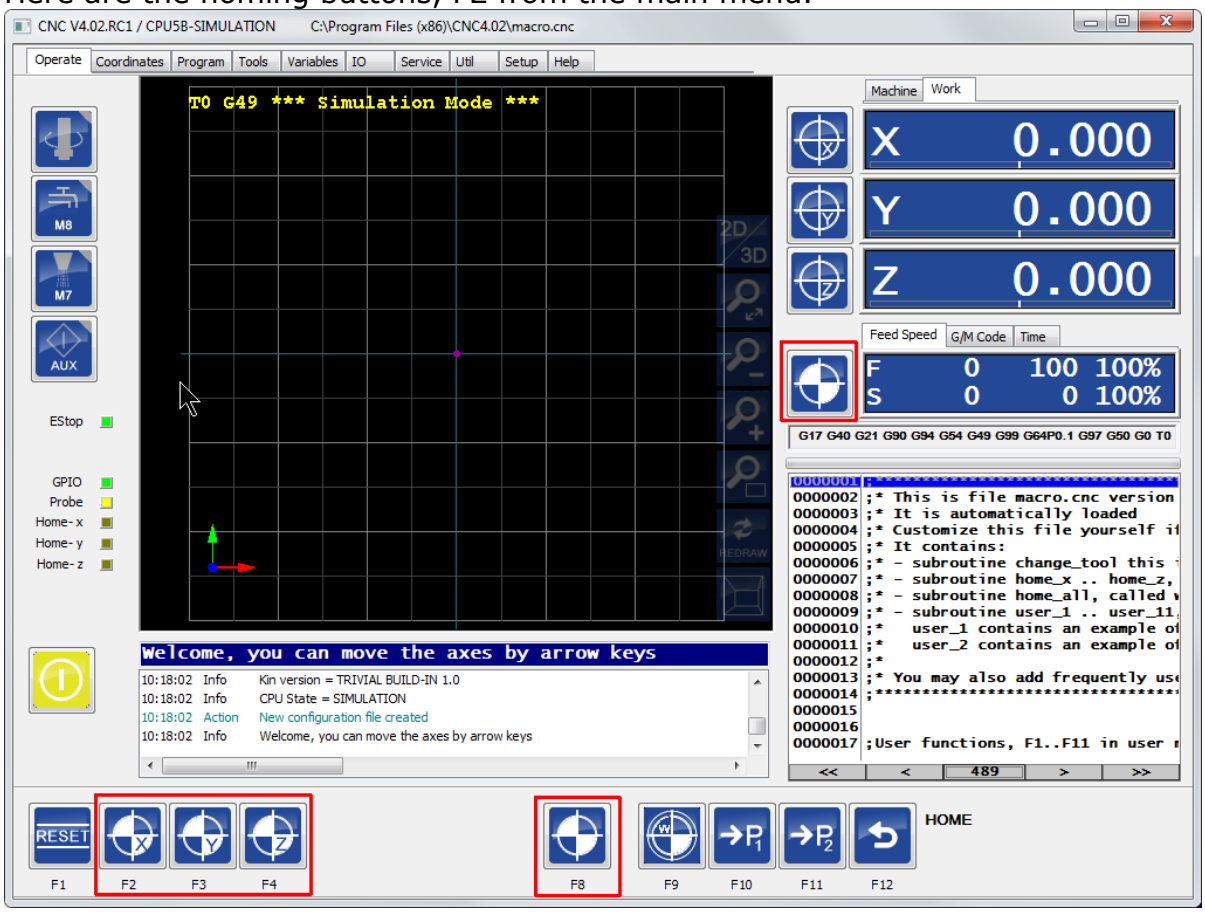
This page allows to calculate the right Feed/Speed for milling.
Chip load is the quantity of material that is removed by one teeth of the milling tool.
This is the most important parameter for calculating the feed given a Speed.

2.10 HOMING AND COORDINATE SYSTEMS

As I am like most people and don't want to read a comprehensive manual, but start right away. So I have written this little tutorial, it explains how to home the machine and use the coordinate systems in a simple way. This part is very important to read, you will have better experience with the machine if you use the coordinate systems the right way!

When your machine is switched on, all axes can be at any position, these positions are unknown by the software. The software however needs to know the position to show a correct graphic in the graph screen and also for preventing damage to your machine by running beyond the machine limits. The process to match the machine position with the software is called homing. Homing can be done either manually or automatic if end of stroke switches are mounted. This tutorial describes homing.

Here are the homing buttons, F2 from the main menu.


CNC V4.02.RC1 / CPU5B-SIMULATION C:\Program Files (x86)\CNC4.02\macro.cnc
Operate Coordinates Program Tools Variables IO Service Util Setup Help
T0 G49 *** Simulation Mode ***
Machine Work
X 0.000
Y 0.000
Z 0.000
Feed Speed G/M Code Time
F 0 100 100%
S 0 0 100%
G17 G40 G21 G90 G94 G54 G49 G99 G64P0.1 G97 G50 G0 T0
M8
M7
AUX
EStop
GPIO
Probe
Home-x
Home-y
Home-z
Welcome, you can move the axes by arrow keys
10:18:02 Info Kin version = TRIVIAL BUILD-IN 1.0
10:18:02 Info CPU State = SIMULATION
10:18:02 Action New configuration file created
10:18:02 Info Welcome, you can move the axes by arrow keys
;* This is file macro.cnc version
;* It is automatically loaded
;* Customize this file yourself
;* It contains:
;* - subroutine change_tool this
;* - subroutine home_x .. home_z,
;* - subroutine home_all, called
;* - subroutine user_1 .. user_11.
;* user_1 contains an example of
;* user_2 contains an example of
;* You may also add frequently us
;User functions, F1..F11 in user
<< < 489 > >>
RESET
HOME
F1 F2 F3 F4 F8 F9 F10 F11 F12

With F2 .. F4 the X..Z axes can be homed individually.
With F8 and the button beside the Feed Speed window, the home sequence can be started to home all axes in a sequence.
F11 is the same as the button besides the %100 feedOverride display.

What happens is that a few subroutines are called. The subroutines are in the macro.cnc file in your EDINGCNC installation folder. They look like this:

```
;Homing per axis
Sub home_x
```

```
   home x
Endsub

Sub home_y
   home y
Endsub

Sub home_z
   home z
Endsub


;Home all axes, uncomment or comment the axes you want.
sub home_all
   gosub home_z
   gosub home_x
   gosub home_y
endsub
```

A good reader has seen that the order of homing is defined by the **home_all** subroutine and can be customized to your own needs.

2.10.1 **Manual homing the machine**

Homing is the first thing you always do after switching on the machine, I recommend making a habit of it. Suppose your machine limits are:

X: +300 mm and -300 mm
Y: +200 mm and -200 mm
Z: +100 mm and 0 (0 is the bottom surface of the bed)

Set the Home velocity to 0 for all axes that have no EOS switch.

Mark a point somewhere on the machine that you want to use as home reference point, let's say X= -200.0mm, which is 100.0mm from the left edge and Y= -150, which is 50.0 mm from the lower edge. For Z we take to bottom of the bed at Z=0 mm. This position x=-250, Y=-150, Z=0 is entered in the Home Position values in the set up screen, this need to be done once.

Using the arrow keys, jog the X,Y axes to the marked position on the bed and move the Z completely up to the surface of the machine. When the machine is at the position press the Home button in the Home submenu, F2-F7 for X-C.

Be sure that you have set the home velocities of the axes to zero, otherwise the axes will start to move. Now click the buttons X, Y, Z, and A if you have an A-axis. That is all, the axes are now homed and the software now knows the machine position.
As a side effect, now also the software limit switches are enabled which protect you from jogging further than the machine can go. Also the Software limit guard is on that will stop a running program when going beyond the limits.

You may also have noticed that the position mode is set to "machine", this is because homing directly affects the machine coordinate system. From this point the machine coordinate system, is not changed any more, it stays as is.

HINT: Move your machine always back to the home position if you are done with the machine. You don't have to move manually to this point next time when you switch back on the machine. You can do a fast move in machine coordinates like this: g53 g0 x0 y0 z0, or first undo the preset (preset dialog, undo preset) and then do a regular G0.
Another possibility to move quickly to the home positions is using g28,
In the variable window set G28 home positions to the same value as the home positions in the set up window. Now you have to type only g28 to go to the home position.

2.10.2 Automatic homing the machine and HomeIsEstop

The machine needs a homing sensor or switch for each axis connected the its home input on the CPU board.

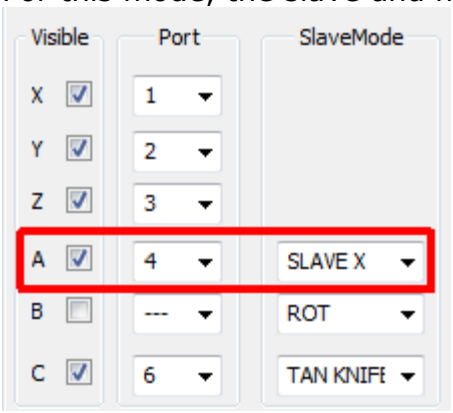
The homing switch is placed at a small distance of the mechanical end of the machine. This distance is needed to ramp down the velocity after the switch is activated.
The sensor should be mounted such that it remains active until the mechanical limit of the machine.

For automatic homing the home velocity needs to be set to another value than zero, use an equal or lower speed than the axis maximum speed. The axis should start to move in the direction where your homing switch is mounted, when it is needed to reverse the direction add a minus sign to the homing velocity. Setup the **HomeInputSenseLevel** correctly. When the home input led's on the IO screen are grey when the input is not activated put a 1 here, when the led's are yellow when the switch is not activated, put a 0. This depends whether you have used normally open or normally closed switch. I recommend normally closed switches here.
Use the homing sub menu to home your axes.

1st Move: The machine first moves until the switch activates, then ramps down and stops.
2nd Move: Then the direction reverses and ramps down when the switch releases.
At the moment of the release of the switch, the position is captured and used to set your machine position correctly.

2.10.3 **Tandem axes homing**

Tandem axes, one main axis has 2 motors, the correct **SlaveMode** is set here. For this mode, the slave and master must each have a homing sensor.



Also modify the **macro.cnc** because this contains the homing sequence, in this example A is slave of X.

```
Sub home_x
   homeTandem X
Endsub
```

And home_a contains nothing:

```
Sub home_a
Endsub
```

Note that sub routine home_all remains unchanged.

For tandem axes these special interpreter commands also exist, use them for testing the sequence if you need, this is what **homeTandem** is doing in separate steps. Use them in **MDI** for testing and understanding.

1. **PrepareTandemHome X**, Both slave and master are moved towards the home sensor. The axes stop when both axes are on the sensor. When one axis reaches the home sensor first, this one is stopped and the other moves further. This movement is done when both axes have reached the sensor.

2. **Home X**, home the X, the slave will just follow. The X home sensor is used. Because the X is already are on the sensor, the move will be towards the machining area **off** the sensor. The position is latched at the moment the sensor de-activates. Then the movement stops and then the correct position is calculated end set for the X.

3. **Home A**, exactly the same, but now the A Home sensor used and the A position is set. When done, A is no longer a slave because the position is different with X. Next command will make de master slave-coupling again.

4. At this point both master and slave have a correct known position but probably different. It can be equalized by moving the slave axis to the position of the A axis. This is done by command **MoveSlaveToMaster** A. The slave will move to the same position as the master. The bridge is set straight and we are done. (If the bridge is not straight, adjust the home positions in the setup).

2.10.4 **Work versus Machine coordinate system and zeroing**

The machine coordinate system does not change, however we want to be able to do the milling of our part anywhere we want on the machine. We will normally use the "work" coordinate system, we can shift it anywhere we want. This can be done with several G-Codes, which are explained in chapter 3, it can also be done using the "preset" button on the operator screen, we'll see this in a minute.

Suppose our g-code file containing the work piece is created with an origin of X=0, Y=0, Z=0. This is because you have drawn your part in a CAD program beginning from these coordinates and then converted to G-Code.
Now you have put your raw material somewhere on the machine, probably not at coordinates X=0, Y=0, Z=0.
By the way, I prefer to define the upper surface of the material as Z=0, such that a negative Z value goes into the material.

Just move to the zero point of the work piece and there press the zero buttons in the operate screen besides the position display.

For the advanced users: The zeroing can also be done using a measuring probe connected to the probe input. An example is provided in the standard macro.cnc file. Under user_1 you find automatic zeroing. Under user_2 you find interactive tool length measurement.

If you want to do it a more advanced way, look at G55 .. G59.3 and also at the G92 variants.

When homing and zeroing is performed, the milling can start:

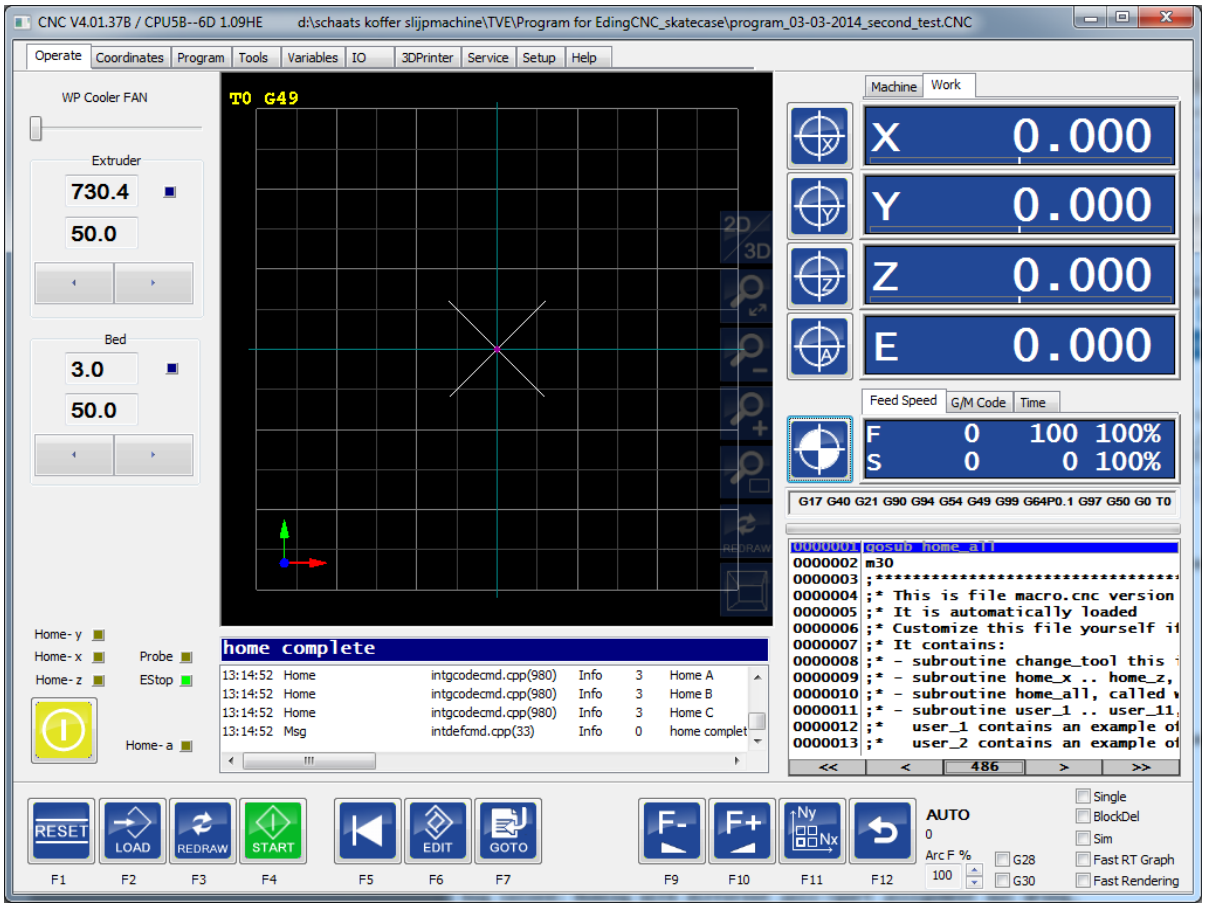
When the program is loaded, go to the graphics screen (Alt-g) and press update preview, you will now see exactly where the part is going to be milled at the surface of you machine bed.

Now press the F4 key or the run button to start milling, go to the graphic screen and switch real time graph on to see what the machine is doing.

That's all for this tutorial, happy milling!

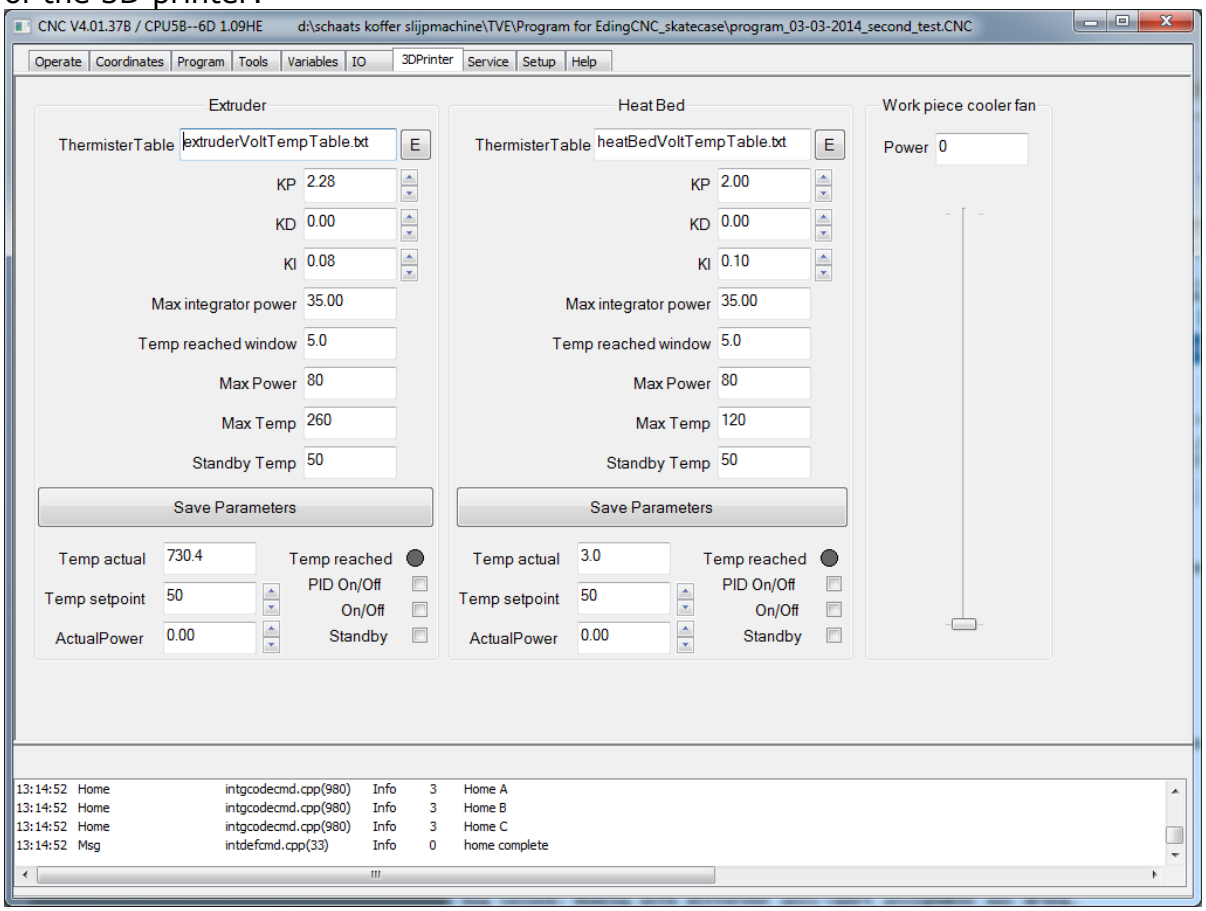
2.11 3D PRINTING

In the setup you can select is3DPrinter at interpreter settings.
This changes the A Axis into an E axis for Extruder.
The operate screen looks slightly different:



On the left side you see controls for the 3D printer:
WP Cooler fan: Controls the speed of the cooler fan that is cooling the work piece. For Extruder and Bed temperature, you see the temperature setting and the actual temperature. The temperature setting can be modified on the fly.

An additional TAB 3D printer shows further controls of the temperature and FAN of the 3D printer:



There are 2 equal parts for Extruder and Heat bed control and a slide to adjust the cooler Fan for the work piece. The temperature is read from analog inputs of the CPU and the Heat and Fan power is controlled by PWM outputs.

The temperature is controlled by a PID controller in software, KP, KD and KI are the PID parameters.
KP: The output power is linear wit KP * temperature difference (set point-actual). By using the KP alone and leaver the KSD and KI zero, you can tune the behavior of the system to keep the temperature stable within a small band. The temperature should be reached in time.
If you get temperature overshoot or oscillation, your KP is too high.

KD (differential), the output power is linear with the derivative of the difference, it looks at changes in temperature and compensates for that. It is not used often but can obtain a more accurate control.

KI is to compensate last differences between set point and actual that cannot be obtained with KP/KD.

Max Integrator power is the max power addition that the integrator function can deliver.
Temp reached window , when the difference between the set point and actual temperature is less than this value, the temperature is considered reached.

Max power is the total maximum power that the system. will give to its PWM output.

Max temp is a safety value when this temperature is reached the power is switched off.
Standby temp, is activate when the system is put in standby mode.

Temp Reached turns green when reached.
PID on/off switch the PID control system OFF.
On/Off when checked you can directly control (to test) the actual power from 1-100%.
Standby activates the standby temperature.

M1 :All heating and Fans off
M104 S :Set extruder temperature, (M104 S50, sets temperature to 50 degree Celsius).
M106 S.. : Work piece cooling FAN ON optionally with S=0-255, for 0-100% PWM
M107 :Work piece FAN off .
M109 S.. :Set extruder temperature and wait until reached.
M143 S.. :Maximum Hot-end temperature to prevent overheating.
M140 S.. :Bed temperature
M143 S.. :Set max extruder temperature
M190 S.. :Set Bed temperature and wait until reached.

All other un-useful or unimplemented special M functions are ignored.

2.12 KEYBOARD SHORTCUTS

Besides the already explained keys for jogging etc, there are a few extra, these are special for pendant builders.

Function	Key
Control + shift + A	Handwheel on A
Control + shift + B	Handwheel on B
Control + shift + C	Handwheel on C
Control + shift + X	Handwheel on X
Control + shift + Y	Handwheel on Y
Control + shift + Z	Handwheel on Z
Alt + shift + A	Select JOG A
Alt + shift + B	Select JOG B
Alt + shift + C	Select JOG C
Alt + shift + X	Select JOG X
Alt + shift + Y	Select JOG Y
Alt + shift + Z	Select JOG Z
Control+Alt+Shift+O	Select JOG Speed Low
Control+Alt+Shift+P	Select JOG Speed Med
Control+Alt+Shift+Q	Select JOG Speed High
Control+Alt+Shift+R	Select JOG Step 0.01
Control+Alt+Shift+S	Select JOG Step 0.1
Control+Alt+Shift+T	Select JOG Step 1
Alt+Shift+P	Start Jog+ selected axis
Alt+Shift+N	Start Jog- selected axis
Alt+Shift+S	Stop Jog
Control D, Control + shift D	Spindle On right, Spindle Off
Control E, Control + shift D	Spindle On left, Spindle Off
Control F, +shift	+Feed, -Feed
Control G, +shift	Run, Pause
Control H,	Home all
Control I	Load g-code file
Control J, + shift J	Jog mode up, jog mode down
Control K	Toggle Flood
Control L	Toggle Mist
Control M	Toggle Aux1
Control N, Control shift N	Handwheel X1, jog continue
Control Alt N	Handwheel X0.1
Control O	Handwheel X10
Control P	Handwheel X100
Control Q	Quit program
Control R	Reset
Control S, Control + shift + S	+Speed, -Speed
Control T	Toggle Single line
Control + B	Toggle Blockdelete
Control U	
Control V, Control shift V	Status tab next, previous
Control W	Toggle Work/Machine coordinates
Control+Alt+S	Speed override 100%
Control+Alt+F	Feed override 100%
Control F1 - control F12	reserved
Control TAB, Control shift TAB	mode
Control F6	toggle MDI
Control 1	Zero x
Control 2	Zero y
Control 3	Zero z
Control 4	Zero a

Control 5	Zero b
Control 6	Zero c
Alt 1,2,3 .. 0, Ctrl+Alt 1,2,3 .. 0	User macro 1 – User Macro 20
Alt F1	Main menu
Alt F2	Home menu
Alt F3	Zero menu
Alt F4	Auto menu
Alt F7	Machine IO menu
Alt F8	Graphics menu
Alt F9	Jog menu
Alt F10	Jog Pad
Alt F11	User 1 menu
Alt F12	User 2 menu

2.13 ZERO TOOL MACRO

User button 1 contains:

```
;Zero tool tip example
Sub user_1
     msg "user_1, Zero Z (G92) using toolsetter"
    (Start probe move, slow)
    f30
    g38.2 z-100
    (Move back to touch point)
    g0 z#5063
    (Set position, the measuring device is 43mm in height, adapt for your measuring device)
    G92 z43
    (move 5 mm above measuring device)
    g91 (incremental distance mode)
    g0 z5
    g90 (absolute distance mode)
    m30
Endsub
```

The idea is to use a flexible position tool setter and put it on top of the workpiece. Start this function and when done, the Z coordinate is set to 0 at the surface of the workpiece.
The feed is set slow F30. A probe move G38.2 is started towards -Z, when the tool setter is touched the position is stored and the movement is stopped. The machine moves exactly to the touch point.
G92 is used with a Z value that specifies the height of you tool setter 43 mm in this case. Change to match your tool setter. An incremental movement is started 5 mm upwards, so you can remove the tool setter. The machine goes back to absolute mode and is done.

WARNING: Take care that you fully understand this macro before using it and that you have adapted it for your own tool setter.

2.14 TOOL MEASUREMENT MACRO

Under user menu button 2 you'll see this:
;Tool length measurement example

```
Sub user_2
   goSub m_tool ;See sub m_tool
Endsub
```

The user 2 button calls subroutine **m_tool**.
This subroutine needs a few values that are stored:

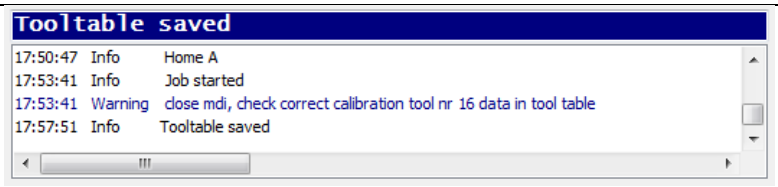
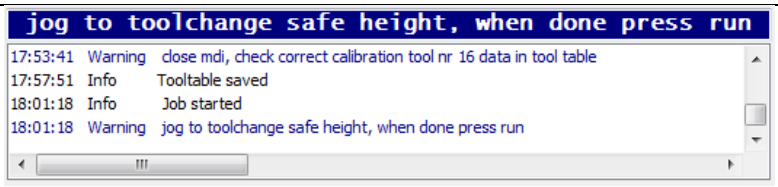
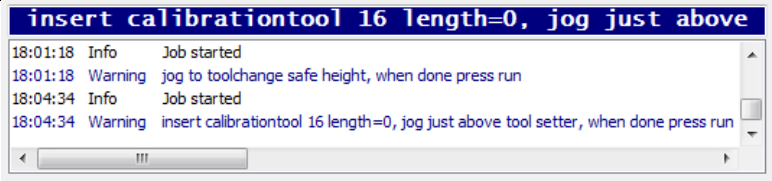
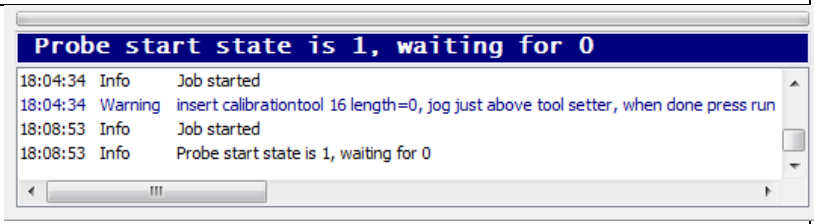
#4996, Z coordinate at tool change safe height
#4997, X coordinate for tool change
#4998, Y coordinate for tool change
#4999, Z coordinate at tool length equals zero or calibration tool height.

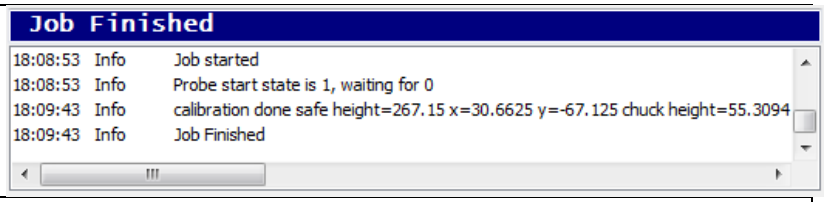
Tool #99 is used as reference tool and should have filled in its tool length before you start. This tool length can be 0 if you use the tool-chuck itself instead of a calibration tool.

The values #4996 .. #4999 are to be determined once. This can be done using the **calibrate_tool_setter** function below. Make sure the machine is homed before you start this.

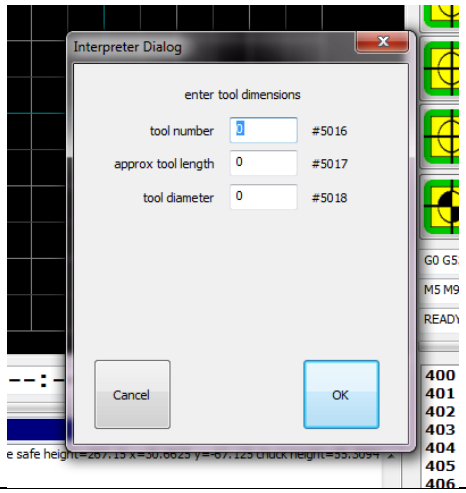
This routine calibrates the safe height, the XY position, the exact height of the tool setter. The positions are stored into persistent variables #4996 - #4999. The positions are used by subroutine m_tool that is under user button 2.

Your action	Machine message
1.Open MDI and type : gosub calibrate_tool_setter Press enter to execute. (Start the calibration procedure)	close mdi, check correct calibration tool nr 16 dat 17:50:47 Info Home Y 17:50:47 Info Home A 17:53:41 Info Job started 17:53:41 Warning close mdi, check correct calibration tool nr 16 data in tool table

2.Close the MDI window using F6, Go to the tools tab and check the tool length of tool 99. For me it is 0 because I use the tool chuck without calibration tool. Press save changes. and go back to the operate tab. (tool 99 is used as calibration tool, set length correctly, normally this would be zero, or if the calibration tool has a defined length, set that length)	
3.The program is still inside subroutine **calibrate_tool_setter.** Press RUN to continue. (Calibration length is set correctly in tool table)	
4.Do what the message says: Jog Z to safe height. In my case this is completely up. Press RUN to continue (This calibrates the safe height)	
5.Do what the messages says: Insert the calibration tool if you have one, or just leave the tool chuck empty. Jog using X, Y, Z just above the tool setter. When done jogging, press RUN.	 The machine will move down to touch the tool setter, The measured tool-chuck height is stored into #4999. Then the Z is moved up to safe height.

(This calibrates the XY position of the tool setter device)	
Calibration DONE **We need to do this only once.** **You need to do this again if you have changed something that influences the calibrated data.**	

When all calibrated, **the user button F2** can be used to measure the tool length.
Make sure the correct tool is loaded before you start.

Press USER BUTTON 2.	The machine moves to safe height. The dialog is shown: 
Type correct values for tool number, tool length and diameter. Press OK.	The machine moves to the correct X,Y The machine moves 10 mm above the tool setter. So make sure the approx tool length above is OK. The machine does the move towards the tool setter, Then calculates and stores the values. Then machine moves Z to safe height.
Tool Length measurement Complete.	

```
Sub calibrate_tool_setter
   warnmsg "close MDI, check correct calibration tool nr 99 data in tool table"
   warnmsg "jog to toolchange safe height, when done press RUN"
   #4996=#5073 ;Store toolchange safe height machine coordinates
   warnmsg "insert calibrationtool 99 length=" #5499 ", jog just above tool setter, when done press RUN"
   ;store x y in non volatile parameters (4000 - 4999)
   #4997=#5071 ;machine pos X
   #4998=#5072 ;machine pos Y
   ;Determine minimum toochuck height and store into #4999
   g38.2 g91 z-20 f30
   #4999=[#5053 - #5499] ;probepos Z - calibration tool length = toolchuck height
   g90
   g0 g53 z#4996
   msg "calibration done safe height="#4996 " X="#4997 " Y="#4998 " Chuck height="#4999
endSub

sub m_tool
   ;Check if toolsetter is calibrated
   if [[#4996 == 0] and [#4997 == 0] and [#4998 == 0] and [#4999 == 0]]
      errmsg "calibrate toolsetter first open mdi, enter gosub calibrate_tool_setter"
   else
      g0 g53 z#4996 ; move to safe z
      dlgmsg "enter tool dimensions" "tool number" 5016 "approx tool length" 5017 "tool diameter" 5018

      if [#5398 == 1];user pressed OK
         if [[#5016 < 1] OR [#5016 > 15]]
            ErrMsg "Tool must be in range of 0 .. 15"
         endif

         ;move to toolsetter coordinates
         g00 g53 x#4997 y#4998
         ;move to 10mm above chuck height + approx tool length + 10
         g00 g53 z[#4999+10+#5017]
                              ;measure tool length and pull 5mm back up
         g38.2 g91 z-20 f30
         g90
         ;back to safe height
         g0 g53 z#4996
         ;Store tool length, diameter in tool table
         #[5400 + #5016] = [#5053-#4999]
         #[5500 + #5016] = #5018
         #[5600 + #5016] = 0 ;Tool X offset is 0
         msg "tool length measured="#[5400 + #5016]" stored at tool "#5016
      endif
   endif
endsub
```

3 Input: the RS274/NGC Language

This chapter describes the input language, RS274/NGC. Overview
The RS274/NGC language is based on lines of code. Each line (also called a "block") may include commands to a machining center to do several different things. Lines of code may be collected in a file to make a program.

A typical line of code consists of an optional line number at the beginning followed by one or more "words." A word consists of a letter followed by a number (or something that evaluates to a number). A word may either give a command or provide an argument to a command. For example, "G1 X3" is a valid line of code with two words. "G1" is a command meaning "move in a straight line at the programmed feed rate," and "X3" provides an argument value (the value of X should be 3 at the end of the move). Most RS274/NGC commands start with either G or M (for miscellaneous). The words for these commands are called "G codes" and "M codes."

The RS274/NGC language has no indicator for the start of a program. The RS274/NGC language has two commands (M2 or M30), either of which ends a program.

3.1 SYSTEM-PARAMETERS/VARIABLES

In the RS274/NGC language view, a machining center maintains an array of 5999 numerical parameters. They can be accessed by #1 .. #5999. The specific parameters with dedicated function are listed in the table below. Other parameters in range of 1..5999 are free to use in your G-Code program.

A simple example of usage:
#1=100 ; assign the value 100 to variable #1
G0[#1] ; use #1 to move to 100

Parameters with specific meaning are listed in this table below.

Parameter number	**Meaning**
1-26	Used for parameters when overriding m-functions. When in the g-code there is e.g. M999 X100 S1000 And you have in your macro.cnc: Sub m999 msg "this is my m999 X="#24" S="#19 End sub Inside the subroutine, the given X and S parameters are at #24 and #19 #1-#26 = A – Z parameter value. Values are negative -1e10 if not provided with m999 in this example.
27-4999	Free to use, note that 4996 – 4999 are used by the tool length measurement function under user button 2.
4000-4999	Free to use, persistent.
5001-5006	POS X – C, interpreter position = work position
5008	Actual TOOL #
5009	Actual TOOL Radius
5010	Actual TOOL Z offset (Length + zDelta)
5011	New tool during tool change
5012	Actual tool X offset (X offset + xDelta)
5013	Actual G43 Z offset (Z offset + zDelta)
5014	Actual G43 X offset (X offset + xDelta)

Parameter number	Meaning
5015 - 5050	Used in tool change sub routine
5051 - 5056	Probe position X – C in machine coordinates
5061 - 5066	Probe position X – C in work coordinates
5067	1 if probe is triggered after G38.2, 0 otherwise.
5068	Actual Probe value
5069	Hand-wheel counter.
5071-5076	POS X – C, interpreter position without offsets = Machine position.
5081-5086	Probe position X – C in joint coordinates
5101-5106	MCA NEG LIMIT X - C
5111-5116	MCA POS LIMIT X - C
5121-5126	HOME X-C
5131-5133	TCA NEG LIMIT X-Z
5141-5143	TCA POS LIMIT X-Z
5150	Active kin type: 1: Trivial 2: 4_AX_ACYLINDER (Y -> A mapping) 3: Virtual C 4-17 : System reserved 18-30: Custom 1 – Custom 12
5151	ZHC is active
5152	1: Spindle is ON, 0:Spindle = OFF
5161 - 5166	G28 home X - C
5181 - 5186	G30 home X - C
5190	G68 Rotation Method (0=OFF, 1=ON)
5191	G68/G51 Rotation point X
5192	G68/G51 Rotation point Y
5193	G68/G51 Rotation point Z
5194	G68 Rotation angle XY
5195	G68 Rotation angle YZ (Not Yet in use)
5196	G68 Rotation angle XZ (Not yet in use)
G5200	G51 Scaling (0=OFF, 1 = ON)
G5204	G51 Scaling factor X
G5205	G51 Scaling factor Y

Parameter number	Meaning
G5206	G51 Scaling factor Z (1.0 always)
5211 - 5216	G92 offset X - C
5220	Coord. System number
5221 - 5226	Coord. System 1 X – C
5241 - 5246	Coord. System 2 X – C
5261 - 5266	Coord. System 3 X – C
5281 - 5286	Coord. System 4 X – C
5301 - 5306	Coord. System 5 X – C
5321 - 5326	Coord. System 6 X – C
5341 - 5346	Coord. System 7 X – C
5361 - 5366	Coord. System 8 X – C
5381 - 5386	Coord. System 9 X – C
5390	Spindle selection 0=M90 1=M91 2=M92
5391 - 5393	Alt spindle Offset X - Z
5394 - 5396	Spindle Speed MAX M90-M91
5230	Reserved for rotation coordinate system 1
5250	Reserved for rotation coordinate system 2
5270	Reserved for rotation coordinate system 3
5290	Reserved for rotation coordinate system 4
5310	Reserved for rotation coordinate system 5
5330	Reserved for rotation coordinate system 6
5350	Reserved for rotation coordinate system 7
5370	Reserved for rotation coordinate system 8
5380	Simulation mode 0=normal 1=Simulation.
5397	Running mode 0=normal 1=rendering Use e.g. if you have G38.2 movements in your macro file because during rendering G38.2 always runs until the given end point which may give incorrect tool measurement results. Always take care that the tool table contains (approx.) correct tool data
5398	Return value for dlgmsg (+1 OK, -1 Cancel)
5399	Return value for M55, M56
5401 – 5499	Tool z offset (Length) Tool 1 – Tool 99
5501 – 5599	Tool diameter Tool 1 – Tool 99
5601 – 5699	Tool x offset (for Turning) Tool 1 – Tool 99

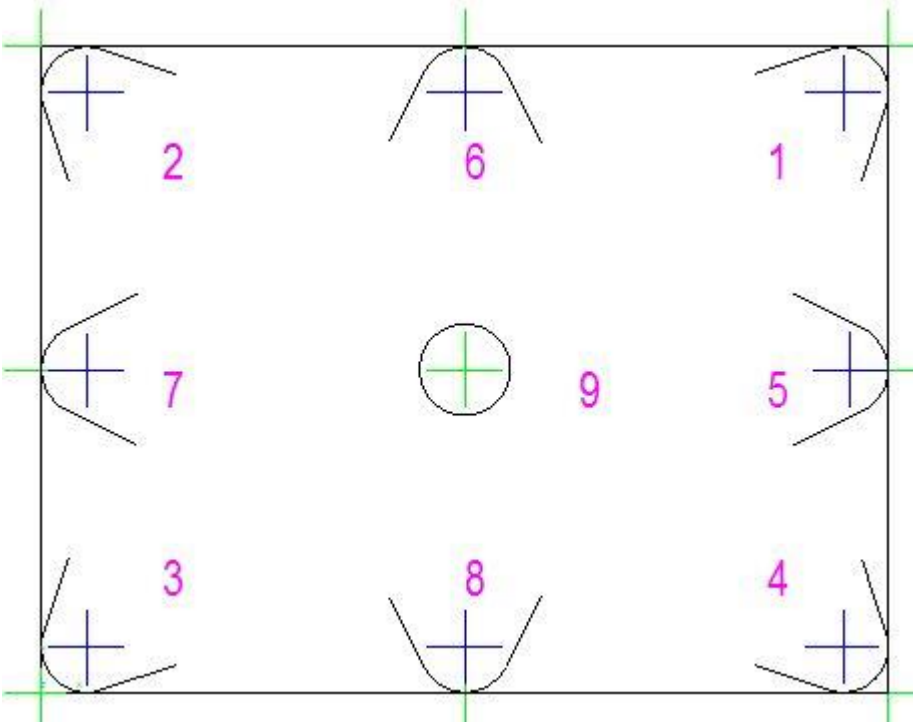
Parameter number	Meaning
5701 – 5799	Tool orientation (for Turning) Tool 1 – Tool 99
	(Currently supported only Tool 0 .. Tool 99)
5801 - 5899	Tool X Delta due to Wear
5901 - 5999	Tool Z Delta due to Wear

3.2 TOOL DATA

Tool ID	zOffset (Length)	xOffset (For turning)	Diameter	orientation
1				1-9
2				1-9
..				..
99				1-9

3.2.1.1 TOOL ORIENTATION FOR LATHES

When the G18 plane (X-Z) is selected, special LATHE tool radius compensation can be used (G41, G42). Depending on the tool orientation and tool radius an extra offset is applied.



The blue crosses show the radius center of the tool.
The green crosses show the controlled point depending on the tool orientation.
For orientation = 9 there is no offset compensation. For orientation = 2, the compensation in X is –tool radius, in Z also –tool radius.

3.3 COORDINATE SYSTEMS

In the RS274/NGC language view, a machining center has an absolute coordinate system and nine program coordinate systems.

You can set the offsets of the nine program coordinate systems using G10 L2 Pn (n is the number of the coordinate system) with values for the axes in terms of the absolute coordinate system.

You can select one of the nine systems by using G54, G55, G56, G57, G58, G59, G59.1, G59.2, or G59.3 . It is not possible to select the absolute coordinate system directly.

You can offset the current coordinate system using G92 or G92.3. This offset will then apply to all nine program coordinate systems. This offset may be cancelled with G92.1 or G92.2.
You can make straight moves in the absolute machine coordinate system by using G53 with either G0 or G1.

Data for coordinate systems is stored in parameters, see the previous section. During initialization, the coordinate system is selected that is specified by parameter 5220. A value of 1 means the first coordinate system (the one G54 activates), a value of 2 means the second coordinate system (the one G55 activates), and so on. It is an error for the value of parameter 5220 to be anything but a whole number between one and nine.

The g-code are described in detail in section 3.6

3.4 FORMAT OF A LINE

A permissible line of input RS274/NGC code consists of the following, in order, with the restriction that there is a maximum (currently 256) to the number of characters allowed on a line.

- An optional line number.
- Any number of words, parameter settings, and comments.

Any input not explicitly allowed is illegal and will cause the Interpreter to signal an error.
Spaces and tabs are allowed anywhere on a line of code and do not change the meaning of the line, except inside comments. This makes some strange-looking input legal. The line "g0x +0. 12 34y 7" is equivalent to "g0 x+0.1234 y7", for example.

Blank lines are allowed in the input. They are to be ignored.
Input is case insensitive.

3.4.1 Line Number

A line number is the letter N followed by an integer (with no sign) between 0 and 99999 written with no more than five digits (000009 is not OK, for example). Line numbers may be repeated or used out of order, although normal practice is to avoid such usage.
Line numbers may also be skipped, and that is normal practice. A line number is not required to be used, but must be in the proper place if used.

3.4.2 Word

A word is a letter other than N followed by a real value.
Words may begin with any of the letters shown in Table 3-2. The table includes N for completeness, even though, as defined above, line numbers are not words. Several letters (I, J, K, L, P, and R) may have different meanings in different contexts.

Letter	Meaning
A	A-axis of machine
D	Tool radius compensation number
F	Feed rate
G	General function (see Table 3-4)
H	Tool length offset index
I	X-axis offset for arcs X offset in G87 canned cycle
J	Y-axis offset for arcs Y offset in G87 canned cycle
K	Z-axis offset for arcs Z offset in G87 canned cycle
L	number of repetitions in canned cycles key used with G10
M	miscellaneous function (see Table 3-6)
N	line number
P	dwell time in canned cycles dwell time with G4 key used with G10

Letter	Meaning
Q	feed increment in G83 canned cycle
R	arc radius, clear_z distance in canned cycle
S	spindle speed
T	tool selection
X	X-axis of machine
Y	Y-axis of machine
Z	Z-axis of machine
A	A-axis of machine
B	B-axis of machine
C	C-axis of machine

A real value is some collection of characters that can be processed to come up with a number. A real value may be an explicit number (such as 341 or -0.8807), a parameter value, an expression, or a unary operation value. Definitions of these follow immediately. Processing characters to come up with a number is called "evaluating". An explicit number evaluates to itself.

3.4.2.1 NUMBER

The following rules are used for (explicit) numbers. In these rules a digit is a single character between 0 and 9.

- A number consists of (1) an optional plus or minus sign, followed by (2) zero to many digits, followed, possibly, by (3) one decimal point, followed by (4) zero to many digits - provided that there is at least one digit somewhere in the number.
- There are two kinds of numbers: integers and decimals. An integer does not have a decimal point in it; a decimal does.
- Numbers may have any number of digits, subject to the limitation on line length. Only about seventeen significant figures will be retained, however (enough for all known applications).
- A non-zero number with no sign as the first character is assumed to be positive.

Notice that initial (before the decimal point and the first non-zero digit) and trailing (after the decimal point and the last non-zero digit) zeros are allowed but not required.
A number written with initial or trailing zeros will have the same value when it is read as if the extra zeros were not there.

Numbers used for specific purposes in RS274/NGC are often restricted to some finite set of values or some to some range of values. In many uses, decimal numbers must be close to integers; this includes the values of indexes (for parameters and carousel slot numbers, for example), M codes, and G codes multiplied by ten. A decimal number which is supposed be close to an integer is considered close enough if it is within 0.0001 of an integer.

3.4.2.2 PARAMETER VALUE

A parameter value is the pound character # followed by a real value. The real value must evaluate to an integer between 1 and 5399. The integer is a parameter number, and the value of the parameter value is whatever number is stored in the numbered parameter.
The # character takes precedence over other operations, so that, for example, "#1+2" means the number found by adding 2 to the value of parameter 1, not the value found in parameter 3. Of course, #[1+2] does mean the value found in parameter 3. The # character may be repeated; for example ##2 means the value of the parameter whose index is the (integer) value of parameter 2.

3.4.2.3 EXPRESSIONS AND BINARY OPERATIONS

An expression is a set of characters starting with a left bracket [and ending with a balancing right bracket]. In between the brackets are numbers, parameter values, mathematical operations, and other expressions. An expression may be evaluated to produce a number. The expressions on a line are evaluated when the line is read, before
anything on the line is executed. An example of an expression is [1 + acos[0] - [#3 ** [4.0/2]]].

Binary operations appear only inside expressions. Nine binary operations are defined. There are four basic mathematical operations: addition (+), subtraction (-), multiplication (*), and division (/). There are three logical operations: non-exclusive or (OR), exclusive or (XOR), and logical and (AND). The eighth operation is the modulus operation (MOD). The ninth operation is the "power" operation (**) of raising the number on the left of the operation to the power on the right.

The binary operations are divided into three groups. The first group is: power. The second group is: multiplication, division, and modulus. The third group is: addition, subtraction, logical non-exclusive or, logical exclusive or, and logical and. If operations are strung together (for example in the expression [2.0 / 3 * 1.5 - 5.5 / 11.0]), operations in the first group are to be performed before operations in the second group and operations in the second group before operations in the third group. If an expression contains more than one operation from the same group (such as the first / and * in the example), the operation on the left is performed first. Thus, the example is equivalent to: [((2.0 / 3) * 1.5) - (5.5 / 11.0)] , which simplifies to [1.0 - 0.5] , which is 0.5.

The logical operations and modulus are to be performed on any real numbers, not just on integers. The number zero is equivalent to logical false, and any non-zero number is equivalent to logical true.

3.4.2.4 UNARY OPERATION VALUE

A unary operation value is either "ATAN" followed by one expression divided by another expression (for example "ATAN[2]/[1+3]") or any other unary operation name followed by an expression (for example "SIN[90]"). The unary operations are: ABS (absolute value), ACOS (arc cosine), ASIN (arc sine), ATAN (arc tangent), COS (cosine), EXP (e raised to the given power), FIX (round down), FUP (round up), LN (natural logarithm), ROUND (round to the nearest whole number), SIN (sine), SQRT (square root), and TAN (tangent). Arguments to unary operations which take angle measures (COS, SIN, and TAN) are in

degrees. Values returned by unary operations which return angle measures (ACOS, ASIN, and ATAN) are also in degrees.

The FIX operation rounds towards the left (less positive or more negative) on a number line, so that FIX[2.8] =2 and FIX[-2.8] = -3, for example. The FUP operation rounds towards the right (more positive or less negative) on a number line; FUP[2.8] = 3 and FUP[-2.8] = -2, for example.

3.4.3 **Parameter Setting**

A parameter setting is the following four items one after the other: (1) a pound character # , (2) a real value which evaluates to an integer between 1 and 5399, (3) an equal sign
= , and (4) a real value. For example "#3 = 15" is a parameter setting meaning "set parameter 3 to 15."

A parameter setting does not take effect until after all parameter values on the same line have been found. For example, if parameter 3 has been previously set to 15 and the line "#3=6 G1 x#3" is interpreted, a straight move to a point where x equals 15 will occur and the value of parameter 3 will be 6.

3.4.4 **Comments and Messages**

Printable characters and white space inside parentheses is a comment. A left parenthesis always starts a comment. The comment ends at the first right parenthesis found thereafter. Once a left parenthesis is placed on a line, a matching right parenthesis must appear before the end of the line. Comments may not be nested; it is an error if a left parenthesis is found after the start of a comment and before the end of the comment.
Here’s an example of a line containing a comment: "G80 M5 (stop motion)". Comments do not cause a machining center to do anything.

A comment contains a message if "MSG," appears after the left parenthesis and before any other printing characters. Variants of "MSG," which include white space and lower case characters are allowed. The rest of the characters before the right parenthesis are considered to be a message. Messages should be displayed on the message display device. Comments not containing messages need not be displayed there.

3.4.5 **Item Repeats**

A line may have any number of G words, but two G words from the same modal group may not appear on the same line.

A line may have zero to four M words. Two M words from the same modal group may
not appear on the same line.

For all other legal letters, a line may have only one word beginning with that letter.

If a parameter setting of the same parameter is repeated on a line, "#3=15 #3=6", for example, only the last setting will take effect. It is silly, but not illegal, to set the same parameter twice on the same line.

If more than one comment appears on a line, only the last one will be used; each of the other comments will be read and its format will be checked, but it will be ignored thereafter. It is expected that putting more than one comment on a line will be very rare.

3.4.6 **Item order**

The three types of item whose order may vary on a line (as given at the beginning of this section) are word, parameter setting, and comment. Imagine that these three types of item are divided into three groups by type.

The first group (the words) may be reordered in any way without changing the meaning of the line.

If the second group (the parameter settings) is reordered, there will be no change in the meaning of the line unless the same parameter is set more than once. In this case, only the last setting of the parameter will take effect. For example, after the line "#3=15 #3=6" has been interpreted, the value of parameter 3 will be 6. If the order is reversed to "#3=6 #3=15" and the line is interpreted, the value of parameter 3 will be 15.

If the third group (the comments) contains more than one comment and is reordered, only the last comment will be used.

If each group is kept in order or reordered without changing the meaning of the line, then the three groups may be interleaved in any way without changing the meaning of the line. For example, the line "g40 g1 #3=15 (foo) #4=-7.0" has five items and means exactly the same thing in any of the 120 possible orders (such as "#4=-7.0 g1 #3=15 g40 (foo)") for the five items.

3.4.7 **Commands and Machine Modes**

In RS274/NGC, many commands cause a machining center to change from one mode to another, and the mode stays active until some other command changes it implicitly or explicitly. Such commands are called "modal". For example, if coolant is turned on, it stays on until it is explicitly turned off. The G codes for motion are also modal. If a G1 (straight move) command is given on one line, for example, it will be executed again on
the next line if one or more axis words are available on the line, unless an explicit command is given on that next line using the axis words or cancelling motion.

"Non-modal" codes have effect only on the lines on which they occur. For example, G4 (dwell) is non-modal.

3.5 MODAL GROUPS

Modal commands are arranged in sets called "modal groups", and only one member of a modal group may be in force at any given time. In general, a modal group contains commands for which it is logically impossible for two members to be in effect at the same time - like measure in inches vs. measure in millimeters. A machining center may be in many modes at the same time, with one mode from each modal group being in effect. The modal groups are shown in Table 3-3.

Table 3-3 Modal Groups

The modal groups for G codes are: group 1 = {G0, G1, G2, G3, G38.2, G76, G80, G81, G82, G83, G84, G85, G86, G87, G88, G89} motion group 2 = {G17, G18, G19} plane selection group 3 = {G90, G91} distance mode group 5 = {G93, G94} feed rate mode group 6 = {G20, G21} units group 7 = {G40, G41, G42} cutter radius compensation group 8 = {G43, G49} tool length offset group 10 = {G98, G99} return mode in canned cycles group 12 = {G54, G55, G56, G57, G58, G59, G59.1, G59.2, G59.3} coordinate system selection group 13 = {G61, G61.1, G64} path control mode group 14 = {G68, G69} XY plane rotation.
The modal groups for M codes are: group 4 = {M0, M1, M2, M30, M60} stopping group 5 = {54, M55, M56, M64, M65, M66} AUX and general purpose I/O group 6 = {M6} tool change group 7 = {M3, M4, M5} spindle turning group 8 = {M7, M8, M9} coolant (special case: M7 and M8 may be active at the same time) group 9 = {M48, M49, M50, M51, M52} enable/disable feed and speed override switches group 10 = {M90, M91, M92, M95, M97} select standard or alternate spindle or touch probe or camera offset, M90=standard. Enable THC = {M20, M21} THC ON | THC OFF (Torch height control) A axis clamp = {M26, M27} Clamp on | clamp off.
In addition to the above modal groups, there is a group for non-modal G codes: group 0 = {G4, G10, G28, G30, G53, G92, G92.1, G92.2, G92.3}

For several modal groups, when a machining center is ready to accept commands, one member of the group must be in effect. There are default settings for these modal groups. When the machining center is turned on or otherwise re-initialized, the default values are automatically in effect.

Group 1, the first group on the table, is a group of G codes for motion. One of these is always in effect. That one is called the current motion mode.

It is an error to put a G-code from group 1 and a G-code from group 0 on the same line if both of them use axis words. If an axis word-using G-code from group 1 is implicitly in effect on a line (by having been activated on an earlier line), and a group 0 G-code that uses axis words appears on the line, the activity of the group 1 G-code is suspended for that line. The axis word-using G-codes from group 0 are G10, G28, G30, and G92.

3.6 G CODES

G codes of the RS274/NGC language are shown in Table 3-4 and described in this Section.
The descriptions contain command prototypes, set in **bold** type.
In the command prototypes, three dots (…) stand for a real value. As described earlier, a real value may be (1) an explicit number, 4, for example, (2) an expression**, [2+2]**, for example, (3) a parameter value, **#88**, for example, or (4) a unary function value, acos[0], for example.
In most cases, if axis words (any or all of **X..., Y..., Z..., A..., B..., C...**) are given, they specify a destination point. Axis numbers are in the currently active coordinate system, unless explicitly described as being in the absolute coordinate system. Where axis words are optional, any omitted axes will have their current value. Any items in the command prototypes not explicitly described as optional are required. It is an error if a required item is omitted.

In the prototypes, the values following letters are often given as explicit numbers. Unless stated otherwise, the explicit numbers can be real values. For example, **G10 L2** could equally well be written **G[2*5] L[1+1]**. If the value of parameter 100 were 2, **G10 L#100** would also mean the same. Using real values which are not explicit numbers as just shown in the examples is rarely useful.

If **L...** is written in a prototype the "…" will often be referred to as the "L number". Similarly the "…" in **H...** may be called the "H number", and so on for any other letter.

3.6.1 Rapid Linear Motion - G0

For rapid linear motion, program **G0 X... Y... Z... A...**, where all the axis words are optional, except that at least one must be used. The G0 is optional if the current motion mode is G0. This will produce coordinated linear motion to the destination point at the current traverse rate (or slower if the machine will not go that fast). It is expected that cutting will not take place when a G0 command is executing.

It is an error if:
• All axis words are omitted.

If cutter radius compensation is active, the motion will differ from the above; see Appendix A. If G53 is programmed on the same line, the motion will also differ.

Table 3-4 G Codes

G Code	Meaning
G0	rapid positioning
G1	linear interpolation
G2	circular/helical interpolation (clockwise)
G3	circular/helical interpolation (counterclockwise)
G4	dwell
G10	coordinate system origin setting
G17	XY-plane selection
G18	XZ-plane selection
G19	YZ-plane selection
G20	inch system selection
G21	millimeter system selection
G28	move to park position 1, setup on variable page
G30	move to park position 2, setup on variable page
G33	Lathe, motion synchronized to spindle
G38.2	straight probe
G40	cancel cutter radius compensation
G41	start cutter radius compensation left
G42	start cutter radius compensation right
G43	tool length offset (plus) , tool X offset for lathe
G49	cancel tool length offset
G53	motion in machine coordinate system
G54	use preset work coordinate system 1
G55	use preset work coordinate system 2
G56	use preset work coordinate system 3
G57	use preset work coordinate system 4
G58	use preset work coordinate system 5
G59	use preset work coordinate system 6
G59.1	use preset work coordinate system 7
G59.2	use preset work coordinate system 8
G59.3	use preset work coordinate system 9
G61	set path control mode: exact path
G61.1	set path control mode: exact stop
G64	set path control mode: continuous
G68	XY rotation
G76	Lathe, threading
G80	cancel motion mode (including any canned cycle)
G81	canned cycle: drilling
G82	canned cycle: drilling with dwell
G83	canned cycle: peck drilling
G84	canned cycle: right hand tapping
G85	canned cycle: boring, no dwell, feed out
G86	canned cycle: boring, spindle stop, rapid out
G87	canned cycle: back boring
G88	canned cycle: boring, spindle stop, manual out
G89	canned cycle: boring, dwell, feed out
G90	absolute distance mode
G91	incremental distance mode
G92	offset coordinate systems and set parameters
G92.1	cancel offset coordinate systems and set parameters to zero

G92.2	cancel offset coordinate systems but do not reset parameters
G92.3	apply parameters to offset coordinate systems
G93	inverse time feed rate mode
G94	units per minute feed rate mode
G98	initial level return in canned cycles
G99	R-point level return in canned cycles

3.6.2 **Linear Motion at Feed Rate - G1**

For linear motion at feed rate (for cutting or not), program **G1 X… Y… Z… A…**, where all the axis words are optional, except that at least one must be used.

The G1 is optional if the current motion mode is G1. This will produce coordinated linear motion to the destination point at the current feed rate (or slower if the machine will not go that fast).

It is an error if:

- All axis words are omitted.

If cutter radius compensation is active, the motion will differ from the above; see Appendix A. If G53 is programmed on the same line, the motion will also differ.

3.6.3 Arc at Feed Rate - G2 and G3

A circular or helical arc is specified using either G2 (clockwise arc) or G3 (counterclockwise arc). The axis of the circle or helix must be parallel to the X, Y, or Z-axis of the machine coordinate system. The axis (or, equivalently, the plane perpendicular to the axis) is selected with G17 (Z-axis, XY-plane), G18 (Y-axis, XZ-plane), or G19 (X-axis, YZ-plane). If the arc is circular, it lies in a plane parallel to the selected plane.

If a line of RS274/NGC code makes an arc and includes rotational axis motion, the rotational axes turn at a constant rate so that the rotational motion starts and finishes when the XYZ motion starts and finishes. Lines of this sort are hardly ever programmed.
If cutter radius compensation is active, the motion will differ from what is described here. See Appendix A.

Two formats are allowed for specifying an arc. We will call these the center format and the radius format. In both formats the G2 or G3 is optional if it is the current motion mode.

3.6.3.1 RADIUS FORMAT ARC

In the radius format, the coordinates of the end point of the arc in the selected plane are specified along with the radius of the arc. Program **G2 X... Y... Z... A... R...** (or use G3 instead of G2). R is the radius. The axis words are all optional except that at least one of the two words for the axes in the selected plane must be used. The R number is the radius. A positive radius indicates that the arc turns through 180 degrees or less, while a negative radius indicates a turn of 180 degrees to 359.999 degrees. If the arc is helical, the value of the end point of the arc on the coordinate axis parallel to the axis of the helix is also specified.

It is not good practice to program radius format arcs that are nearly full circles or are semicircles (or nearly semicircles) because a small change in the location of the end point will produce a much larger change in the location of the center of the circle (and, hence, the middle of the arc). The magnification effect is large enough that rounding error in a number can produce out-of-tolerance cuts. Nearly full circles are outrageously bad, semicircles (and nearly so) are only very bad. Other size arcs (in the range tiny to 165 degrees or 195 to 345 degrees) are OK.

Here is an example of a radius format command to mill an arc: **G17 G2 x 10 y 15 r 20 z 5**.

That means to make a clockwise (as viewed from the positive Z-axis) circular or helical arc whose axis is parallel to the Z-axis, ending where X=10, Y=15, and Z=5, with a radius of 20. If the starting value of Z is 5, this is an arc of a circle parallel to the XY-plane; otherwise it is a helical arc.

3.6.3.2 CENTER FORMAT ARC

In the center format, the coordinates of the end point of the arc in the selected plane are
specified along with the offsets of the center of the arc from the current location. In this format, it is OK if the end point of the arc is the same as the current point. It is an error if:

- When the arc is projected on the selected plane, the distance from the current point to the center differs from the distance from the end point to the center by more than 0.0002 inch (if inches are being used) or 0.002 millimeter (if millimeters are being used).

When the XY-plane is selected, program **G2 X... Y... Z... A... I... J...** (or use G3 instead of G2). The axis words are all optional except that at least one of X and Y must be used. I and J are the offsets from the current location (in the X and Y directions, respectively) of the center of the circle. I and J are optional except that at least one of the two must be used. It is an error if:

- I and J are both omitted.

When the XZ-plane is selected, program **G2 X... Y... Z... A... I... K...** (or use G3 instead of G2). The axis words are all optional except that at least one of X and Z must be used. I and K are the offsets from the current location (in the X and Z directions, respectively) of the center of the circle. I and K are optional except that at least one of the two must be used. It is an error if:

- I and K are both omitted.

When the YZ-plane is selected, program **G2 X... Y... Z... A... B... C... J... K...** (or use G3 instead of G2). The axis words are all optional except that at least one of Y and Z must be used. J and K are the offsets from the current location (in the Y and Z directions, respectively) of the center of the circle. J and K are optional except that at least one of the two must be used. It is an error if:

- J and K are both omitted.

Here is an example of a center format command to mill an arc:
G17 G2 x10 y16 i3 j4 z9.

That means to make a clockwise (as viewed from the positive z-axis) circular or helical arc whose axis is parallel to the Z-axis, ending where X=10, Y=16, and Z=9, with its center offset in the X direction by 3 units from the current X location and offset in the Y direction by 4 units from the current Y location. If the current location has X=7, Y=7 at the outset, the center will be at X=10, Y=11. If the starting value of Z is 9, this is a circular arc; otherwise it is a helical arc. The radius of this arc would be 5.

In the center format, the radius of the arc is not specified, but it may be found easily as the distance from the center of the circle to either the current point or the end point of the arc.

3.6.4 **Dwell - G4**

For a dwell, program G4 P… . This will keep the axes unmoving for the period of time in seconds specified by the P number. It is an error if:

- the P number is negative.

3.6.5 **Set Coordinate System Data -G10**

To set the coordinate values for the origin of a coordinate system, program **G10 L2 P … X… Y… Z… A…**, where the P number must evaluate to an integer in the range 1 to 9 (corresponding to G54 to G59.3) and all axis words are optional. The coordinates of the origin of the coordinate system specified by the P number are reset to the coordinate values given (in terms of the absolute coordinate system). Only those coordinates for which an axis word is included on the line will be reset.

It is an error if:

- the P number does not evaluate to an integer in the range 1 to 9.

If origin offsets (made by G92 or G92.3) were in effect before G10 is used, they will continue to be in effect afterwards.

The coordinate system whose origin is set by a G10 command may be active or inactive at the time the G10 is executed.
Example:**G10 L2 P1 x 3.5 y 17.2** sets the origin of the first coordinate system (the one selected by G54) to a point where X is 3.5 and Y is 17.2 (in absolute coordinates). The Z coordinate of the origin (and the coordinates for any rotational axes) are whatever those coordinates of the origin were before the line was executed.

G10 L20 P.. X.. Y.. Z.. A..

Set coordinate system given by P number relative to actual machine position. Working is similar to G92. Jog to any position, then apply e.g. G10 L20 P1 X0 Y0 to set G54 coordinate system zero point at current machine position.

3.6.6 **Plane Selection - G17, G18, and G19**

Program **G17** to select the XY-plane, **G18** to select the XZ-plane, or **G19** to select the YZ-plane.

3.6.7 **Length Units - G20/G21 and G70/G71**

Program G20 to use inches for length units. Program G21 to use millimeters.

It is usually a good idea to program either G20 or G21 near the beginning of a program before any motion occurs, and not to use either one anywhere else in the program. It is the responsibility of the user to be sure all numbers are appropriate for use with the current length units. G70/G71 is added for CAM software compatibility.

3.6.8 **Return to Home - G28 and G30**

Two home positions are defined (by parameters 5161-5166 for G28 and parameters
5181-5186 for G30). The parameter values are in terms of the absolute coordinate system, but are in unspecified length units.
To return to home position by way of the programmed position, program **G28 X... Y... Z... A...** (or use G30). All axis words are optional. The path is made by a traverse move from the current position to the programmed position, followed by a traverse move to the home position. If no axis words are programmed, the intermediate point is the current point, so only one move is made.

3.6.9 **G33, G33.1 Spindle-Synchronized Motion**

For spindle-synchronized motion in one direction, program
G33 X… Y… Z… K… where K gives the distance moved in XYZ for each revolution of the spindle.

For G33 the software performs this:

1. Start a synchronized move with spindle with K feed per revolution. Assumed is that the spindle already runs (M3).
2. Done.

For G33.1 the software performs:

1. Start a synchronized move with spindle with K feed per revolution. Assumed is that the spindle is running (M3).
2. Wait until this motion is done.
3. Reverse spindle direction.
4. Move back to original position where we were before the G33.1.
5. Done

All the axis words are optional, except that at least one must be used.
It is an error if:

- all axis words are omitted.
- the spindle is not turning when this command is executed.
- the requested linear motion exceeds machine velocity limits due to the spindle speed.

3.6.10 Straight Probe - G38.2

3.6.10.1 THE STRAIGHT PROBE COMMAND

Program **G38.2 X... Y... Z... A...** to perform a straight probe operation. The rotational axis words are allowed, but it is better to omit them. If rotational axis words are used, the numbers must be the same as the current position numbers so that the rotational axes do not move. The linear axis words are optional, except that at least one of them must be used. The tool in the spindle must be a probe.
It is an error if:

- the current point is less than 0.254 millimeter or 0.01 inch from the programmed point.
- G38.2 is used in inverse time feed rate mode,
- any rotational axis is commanded to move,
- no X, Y, or Z-axis word is used.

In response to this command, the machine moves the controlled point (which should be at the end of the probe tip) in a straight line at the current feed rate toward the programmed point. If the probe trips, the probe is retracted slightly from the trip point at the end of command execution. If the probe does not trip even after overshooting the programmed point slightly, an error is signaled.

After successful probing, parameters 5061 to 5066 will be set to the program coordinates of the location of the controlled point at the time the probe tripped. The variables 5051 to 5056 will contain the machine coordinates. Useful for measuring tools in absolute machine positions. G53 G38.2 will move in machine coordinates.

3.6.10.2 USING THE STRAIGHT PROBE COMMAND

Using the straight probe command, if the probe shank is kept nominally parallel to the Z-axis (i.e., any rotational axes are at zero) and the tool length offset for the probe is used, so that the controlled point is at the end of the tip of the probe:

- without additional knowledge about the probe, the parallelism of a face of a part to the XY-plane may, for example, be found.
- if the probe tip radius is known approximately, the parallelism of a face of a part to the YZ or XZ-plane may, for example, be found.
- if the shank of the probe is known to be well-aligned with the Z-axis and the probe tip radius is known approximately, the center of a circular hole, may, for example, be found.
- if the shank of the probe is known to be well-aligned with the Z-axis and the probe tip radius is known precisely, more uses may be made of the straight probe command, such as finding the diameter of a circular hole.

If the straightness of the probe shank cannot be adjusted to high accuracy, it is desirable to know the effective radii of the probe tip in at least the +X, -X, +Y, and -Y directions. These quantities can be stored in parameters either by being included in the parameter file or by being set in an RS274/NGC program.
Using the probe with rotational axes not set to zero is also feasible. Doing so is more complex than when rotational axes are at zero, and we do not deal with it here.

3.6.10.3 EXAMPLE CODE

As a usable example, the code for finding the center and diameter of a circular hole is shown in Table 3-5. For this code to yield accurate results, the probe shank must be well-aligned with the Z-axis, the cross section of the probe tip at its widest point must be very circular, and the probe tip radius (i.e., the radius of the circular cross section) must be known precisely. If the probe tip radius is known only approximately (but the other conditions hold), the location of the hole center will still be accurate, but the hole diameter will not.

In Table 3-5, an entry of the form <description of number> is meant to be replaced by an actual number that matches the description of number. After this section of code has executed, the X-value of the center will be in parameter 1041, the Y-value of the center in parameter 1022, and the diameter in parameter 1034. In addition, the diameter parallel to the X-axis will be in parameter 1024, the diameter parallel to the Y-axis in parameter 1014, and the difference (an indicator of circularity) in parameter 1035. The probe tip will be in the hole at the XY center of the hole.

The example does not include a tool change to put a probe in the spindle. Add the tool change code at the beginning, if needed.

Table 3-5 Code to Probe Hole

```
N010 (probe to find center and diameter of circular hole)
N020 (This program will not run as given here. You have to)
N030 (insert numbers in place of <description of number>.)
N040 (Delete lines N020, N030, and N040 when you do that.)
N050 G0 Z <Z-value of retracted position> F <feed rate>
N060 #1001=<nominal X-value of hole center>
N070 #1002=<nominal Y-value of hole center>
N080 #1003=<some Z-value inside the hole>
N090 #1004=<probe tip radius>
N100 #1005=[<nominal hole diameter>/2.0 - #1004]
N110 G0 X#1001 Y#1002 (move above nominal hole center)
N120 G0 Z#1003 (move into hole - to be cautious, substitute G1 for G0
here)
N130 G38.2 X[#1001 + #1005] (probe +X side of hole)
N140 #1011=#5061 (save results)
N150 G0 X#1001 Y#1002 (back to center of hole)
N160 G38.2 X[#1001 - #1005] (probe -X side of hole)
N170 #1021=[[#1011 + #5061] / 2.0] (find pretty good X-value of hole
center)
N180 G0 X#1021 Y#1002 (back to center of hole)
N190 G38.2 Y[#1002 + #1005] (probe +Y side of hole)
N200 #1012=#5062 (save results) N210 G0 X#1021 Y#1002 (back to
center of hole)
N220 G38.2 Y[#1002 - #1005] (probe -Y side of hole)
N230 #1022=[[#1012 + #5062] / 2.0] (find very good Y-value of hole
center)
```

```
N240 #1014=[#1012 - #5062 + [2 * #1004]] (find hole diameter in Y-
direction)
N250 G0 X#1021 Y#1022 (back to center of hole)
N260 G38.2 X[#1021 + #1005] (probe +X side of hole)
N270 #1031=#5061 (save results)
N280 G0 X#1021 Y#1022 (back to center of hole)
N290 G38.2 X[#1021 - #1005] (probe -X side of hole)
N300 #1041=[[#1031 + #5061] / 2.0] (find very good X-value of hole
center)
N310 #1024=[#1031 - #5061 + [2 * #1004]] (find hole diameter in X-
direction)
N320 #1034=[[#1014 + #1024] / 2.0] (find average hole diameter)
N330 #1035=[#1024 - #1014] (find difference in hole diameters)
N340 G0 X#1041 Y#1022 (back to center of hole)
N350 M2 (that's all, folks)
```

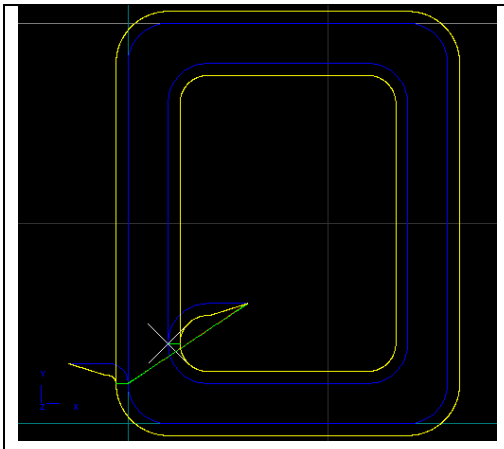
3.6.11 **Cutter Radius Compensation - G40, G41, G41.1, G42, G42.1**

To turn cutter radius compensation off, program G40. It is OK to turn compensation
off when it is already off. Cutter radius compensation may be performed only if the XY-plane is active.
To turn cutter radius compensation on **left** (i.e., the cutter stays to the left of the programmed path when the tool radius is positive), program **G41 D...** . To turn cutter radius compensation on **right** (i.e., the cutter stays to the right of the programmed path when the tool radius is positive), program **G42 D...** . The D word is optional; if there is no D word, the radius of the tool currently in the spindle will be used. If used, the D number should normally be the slot number of the tool in the spindle, although this is not required. It is OK for the D number to be zero; a radius value of zero will be used.
It is an error if:

- the D number is not an integer, is negative or is larger than the number of carousel slots,
- the XY-plane is not active or for turning the ZX plane is not active,
- cutter radius compensation is commanded to turn on when it is already on.

The behavior of the machining center when cutter radius compensation is on is described in Appendix A.
With G41.1 D… is the same as G41 D… except now the D number is not a tool number but a tool diameter.
With G42.1 D… is the same as G42 D… except now the D number is not a tool number but a tool diameter.

3.6.11.1 EXAMPLE CODE FOR MILLING



This example mills out a rectangular object from the outside and inside.
On the outside we use G42, tool radius compensation right and for the inside G41, tool radius compensation left is used.
For both contours a tool-radius-compensation entry move is programmed consisting of a line which must be longer than the tool-radius used and a circle, of which also the radius is bigger than the tool.

By the way, all arc radii should be bigger than the tool radius. If you have inside corners, there should be always an arc, so that the tool fits.

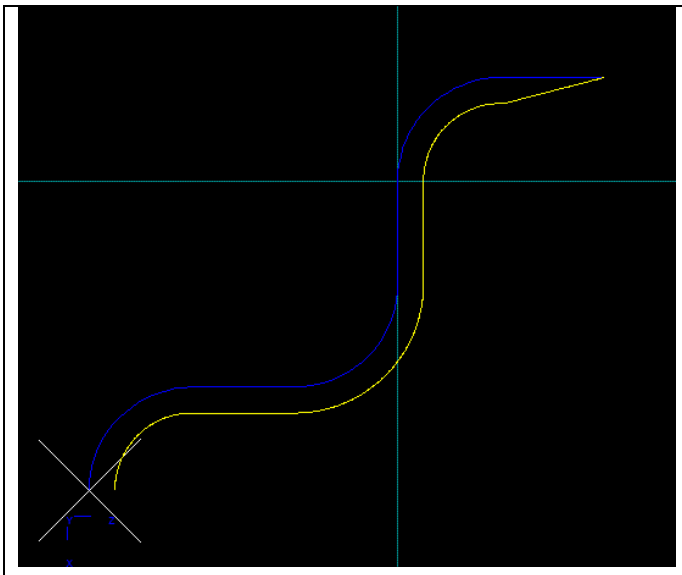
```
g0 z3
g0 x-15 y15
f500
/g42.1 D6
g1 x-5      (cutter comp entry move 1)
g2 x0 y10 r5 (cutter comp entry move 2)
g1 z-3      (plunge down)
g3 x10 y0 r10
g1 x70
g3 x80 y10 r10
g1 y90
g3 x70 y100 r10
g1 x10
g3 x0 y90 r10
g1 x0 y10
/g40
g0 z3
g0 x30 y30
/g41.1 d6
g1 x20
g3 x10 y20 r10
g1 z-3
g3 x20 y10 r10
g1 x60
g3 x70 y20 r10
g1 y80
g3 x60 y90 r10
```

The G42, G41 and G40 codes are programmed with a / (block delete sign) in front. This makes it easy to debug tool comp programs. The program is loaded with block delete on, this is the blue curve.
Then the program is run with block delete off resulting in the yellow curve.

It is clear to see what the entry move does.

g1 x20 g3 x10 y80 r10 g1 y20 /g40 g0 z3 m30	

3.6.11.2 EXAMPLE CODE FOR TURNING

	The movement starts at the right upper corner. The blue line is the programmed contour. The yellow is the contour with tool-radius compensation G41. The first G1 line is the tool comp entry move. You can get this figure by putting a / character in front of the G41/G40 codes. The load the program with block delete on and execute it with block delete off. With block delete on the tool comp is skipped.
(Diameter programming) (Use R word for Arcs) g0 x-20 z20 /g41.1 d5 g1 x-20 z10 g3 x0 z0 r10 g1 x20 g2 x40 z-10 r10 g1 z-20 g3 x60 z-30 r10 /g40 m30	(Radius programming) (Use R word for arc's) g0 x-10 z20 /g41.1 d5 g1 x-10 z10 g3 x0 z0 r10 g1 x10 g2 x20 z-10 r10 g1 z-20 g3 x30 z-30 r10 /g40 m30
(Diameter programming) (Use I,K programming for arc's) g0 x-20 z20 /g41.1 d5 g1 x-20 z10 g3 x0 z0 i10 k0 g1 x20 g2 x40 z-10 i0 k-10 g1 z-20 g3 x60 z-30 i10 k0 /g40 m30	(Radius programming) (Use I,K programming for arc's) g0 x-10 z20 /g41.1 d5 g1 x-10 z10 g3 x0 z0 i10 k0 g1 x10 g2 x20 z-10 i0 k-10 g1 z-20 g3 x30 z-30 i10 k0 /g40

	m30

3.6.12 **Tool Length Offsets - G43, G43 H, G43.1, and G49**

A. To use the tool offset of the tool in the spindle use **G43.**
This assures that always the tool length of the tool in spindle is compensated.

B. To use a tool length offset from the tool table, program **G43 H...**, where the H number is the desired index in the tool table. (H = 1 - 99)

C. To use dynamic tool compensation (not from the tool-table), use G43.1 I.. K.. where I.. gives the tool X offset (turning) and K.. gives the tool Z offset (for turning and milling)

Warning: If you use option B or C, the tool-length compensation will **not** adapt to the new tool after M6T...

To have no tool length offset compensation, program **G49**

#5401 - #5499 is the tool-length of tool 1-99
#5501 - #5599 is the tool-diameter of tool 1-99
#5601 - #5699 is the tool-xoffset (width for turning) offset.

The variables can be modified runtime (in the G-Code file) if needed to compensate for tool-wear.

3.6.13 **Scaling G50/G51**

G50 scaling off.

UNIFORM Scaling
G51 P.. I.. J..
P is scaling factor.

NON UNIFORM Scaling (X,Y different, only applicable when NO Arcs)
G51 X.. Y.. I.. J..
X is scaling factor for X coordinates.
Y is scaling factor for Y coordinates.

I is X coordinate scaling point
J is Y coordinate scaling point

When used in combination with **G68**, the rotation point and scaling point are the same.

3.6.14 **Move in Absolute Coordinates - G53**

For linear motion to a point expressed in absolute coordinates, program **G1 G53 X… Y… Z… A…** (or use G0 instead of G1), where all the axis words are optional, except that at least one must be used. The G0 or G1 is optional if it is the current motion mode. G53 is not modal and must be programmed on each line on which it is intended to be active. This will produce coordinated linear motion to the programmed point. If G1 is active, the speed of motion is the current feed rate (or slower if the machine will not go that fast). If G0 is active, the speed of motion is the current traverse rate (or slower if the machine will not go that fast).

It is an error if:

- G53 is used without G0 or G1 being active,
- G53 is used while cutter radius compensation is on.

3.6.15 **Select Coordinate System - G54 to G59.3**

To select coordinate system 1, program G54, and similarly for other coordinate systems. The system-number-G-code pairs are: (1-G54), (2-G55), (3-G56), (4-G57), (5-G58), (6-G59), (7-G59.1), (8-G59.2), and (9-G59.3).

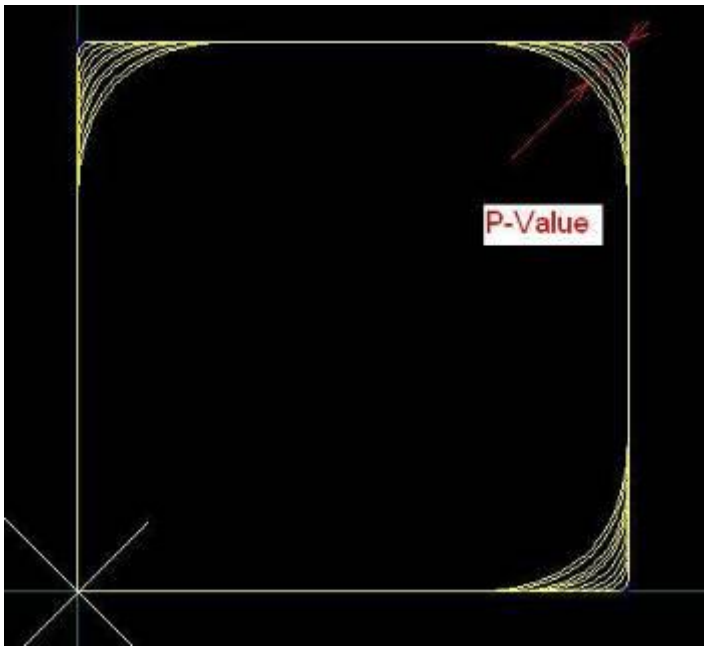
It is an error if:

- one of these G-codes is used while cutter radius compensation is on.

3.6.16 **Set Path Control Mode - G61, and G64**

Ideally one would like to have constant speed during the work and maximum accuracy. This ideal is **physically** not possible, similar as driving a racecar on curvy roads is not possible with constant velocity. It is simply to understand that it is needed to brake in the curves and accelerate on the straight roads. If you would do the same speed in the curves as on the straight road, the car will fly out of the curve and get an accident. With a CNC machine it is the same, constant speed and accuracy together are physically not possible. It would require infinite acceleration to go without ramping down from one direction to another. So a compromise has to be made. There are various options to choose the optimum between accuracy and constant speed.

G61 puts the machining center into exact path mode, In G61, the motion velocity between motion segments goes to zero, the end position in corners is exactly reached, use this if you require maximum accuracy. When a work piece consists of many small lines this gives a quite vibrating machine because of the continuous acceleration-deceleration-stop behavior. More practical is to use G64, see below.
G64 P.. Q.. R.. F.. for continuous velocity mode. In G64, subsequent moves are blended, when previous move starts to decelerate and reaches a velocity such that the specified accuracy isn't violated, the next move starts to accelerate, the two motions are added. The result is smooth motion with highest constant speed. The corners however are rounded. The P value specifies the distance reached to the corner while blending. The next move is blended with current such that the tool path remains no more than P from the corner. The figure below is a rectangle of 10x10 milled with F 2000. This is done with P values from 0.1 to 1, you can see the impact. This gives the best compromise between accuracy and smooth motion.



The optional **P** parameter (Only with G64) controls the max amount of rounding.

The optional **Q** parameter activates an embedded line simplification algorithm, which tries to combine short lines and make one longer line. The optional **Q** parameter gives the tolerance used in the algorithm.

The **R** parameter is the look-ahead feed angle, when subsequent lines/arcs have an angle together less than this value, the trajectory-generator will accelerate through over these segments and this way optimizes the production time. With R0, LAF is switched off, this may be required when milling Elastic materials. A lower but more constant cutting speed will be achieved.

The new **F** parameter defines the value of the Accel/Decel filter placed behind LAF. LAF with high R values (Angle) can cause acceleration spikes because LAF will travel through corners without stopping. The F parameter filters the trajectory generated by LAF and takes care the acceleration is never violated. F1 will give a RAMP time that matches with the max velocity / max acceleration from the setup and so F1 and maximum speed will lead to the max max acceleration that is allowed. Smaller F values filter less. This can be used if the milling velocity is less than the max velocity of the machine. Example:
Max velocity in the setup is 200 and used milling velocity is F6000 (100 mm/s), Then F0.5 is safe to use. So now you can do this G64 R100 F1 and you will not get acceleration spikes but very smooth an fast movement. The price to pay is corner rounding which is depending on the max acceleration of the machine, the higher the max acceleration, the less corner rounding.
A good application example is with milling rubber or similar flexible material, the milling speed must be constant otherwise more material is removed in the corners due to the lower speed and the flexibility of the rubber. The F1 in combination with R100 will give constant and high speed for good milling surface quality of the rubber.

To make a move from stand still we need to accelerate, then have a certain cruising speed and after decelerate. Short moves typically never reach the requested velocity the accelerate and then at half the distance the decelerate.

This table shows the ramp-up / ramp-down distance.

Velociy	Feed	Accel	Distance
48	2880	120	19.20
30	1800	120	7.50
25	1500	120	5.21
15	900	120	1.88
12	720	120	1.20
10	600	120	0.83
9	540	120	0.68
8	480	120	0.53
7	420	120	0.41
6	360	120	0.30
5	300	120	0.21
4	240	120	0.13
3	180	120	0.08

If your machine has higher accelerations which requires bigger motors and light construction, also higher milling velocities are possible. The values given here are for a moderate hobby machine. This illustrates that when the G-Code file

exists of small segments, e.g. 0.08 mm, that with an acceleration of 120 a feed can be reached of 180 mm/minute at most.

The Q and P parameter perform the same function as explained with G64 at the previous page.

Look Ahead feed

To explain this, I will compare a running CNC machine again with driving a race car.

The road maximum velocity signs have to be obeyed and you have to drive your car exactly over the white line in the middle of the road. You will try to reach the maximum allowed velocity where possible. When you see a curve coming up ahead, you will brake so that you will not drift off the road. You will try to look ahead as far as you can see and you take care that you can stop in time if the road suddenly stops.

When you would maintain your speed in sharp curves, you will drift off the road resulting possibly into a car accident. When the road has many short curves, then you will not be able to reach the desired speed. The more PS you have in the car, the higher speed you will reach because you can accelerate faster.

I think this is a good comparison with a CNC machine, the same issues apply. A machine cannot suddenly change velocity, to reach a velocity the motors must accelerate first for a certain time to reach the velocity.

LAF behaves like the ideal racecar driver, it will reach the highest possible velocity without violating the maximum motor accelerations.

There is one additional problem while running CNC programs, some programs consists of short line pieces. When the line pieces connect tangentially (are in line), then LAF will accelerate through over the lines, reaching the maximum allowed speed. Without LAF the speed would not be reached.

The angle to which LAF considers the segments in line is a setup parameter. The theoretical ideal value would be very small, so that no acceleration value occurs. More practical values are in the range of 1 to 4 degrees, the experience learns that most machines can handle acceleration spikes up to a certain limit.

The value can be set up to 180 degrees in this case you must know what you are doing, it can be useful during e.g. foam cutting wing profiles. Be aware however that if the curve contains real sharp angles that step pulse loss may be the result when using large minimum LAF angles.

In practice we have seen that milling times of complex 3D work pieces can be done in 50% of the time compared to competitors who do not have LAF.

With G64 R.. the LAF angle can be changed in the g-code file, see explanation of G64 and G61.

3.6.17 Coordinate system rotation G68

G68 R.. X.. Y..

R Rotation angle in degrees, positive is counter-clockwise, negative is clockwise.
X Y Rotation point in current coordinate system.

3.6.18 Threading (Lathe) – G76

G76 P- Z- I- J- R- K- Q- H- E- L-

P Pitch
Z driveline endpoint
I Outside thread diameter, always positive.
J First cut is J beyond I, always positive.
R Depth regression, use 1.0 for constant cutting depths or leave parameter away.
K Full thread depth beyond thread peak, always positive.
Q Compound slide angle, typical 30.
H Additional spring passes at full depth, use 0 for none.
E Taper distance along drive line.
L Taper place, none, enter, exit, both.

;Create a thread from z=20 to z=10, outside diameter=15, inside diameter=14, 10 passes.
G0 X20 Z20
G76 P1.0 Z10 I15 J0.1 K1.0

It is an error if:

- The active plane is not the ZX plane
- Other axis words, such as X- or Y-, are specified
- The R- degression value is less than 1.0.
- All the required words are not specified
- P-, J-, K- or H- is negative
- E- is greater than half the drive line length

The "drive line" is a safe line outside the thread material. The "drive line" goes from the initial location to the Z- value specified with G76. The Z extent of the thread is the same as the drive line.

The "thread pitch", or distance per revolution, is given by the P- value.

The "thread peak" is given by the I- value, which is an offset from the drive line. Negative I values indicate external threads, and positive I values indicate internal threads. Generally the material has been turned to this size before the G76 cycle.

The “initial cut depth” is given by the J- value. The first threading cut will be J beyond the “thread peak” position. J- is positive, even when I- is negative.

The “full thread depth” is given by the K- value. The final threading cut will be K beyond the “thread peak” position. K- is positive, even when I- is negative.

The “depth degression” is given by the R- value. R1.0 selects constant depth on successive threading passes. R2.0 selects constant area. Values between 1.0 and 2.0 select decreasing depth and increasing area. Values above 2.0 select decreasing area. Beware that unnecessarily high degression values will cause a large number of passes to be used.

The “compound slide angle” Q- is the angle (in degrees) describing to what extent successive passes should be offset along the drive line. This is used to cause one side of the tool to remove more material than the other. A positive Q value causes the leading edge of the tool to cut more heavily. Typical values are 29, 29.5 or 30.

The number of “spring passes” is given by the H- value. Spring passes are additional passes at full thread depth. If no additional passes are desired, program H0.

Tapered entry and exit moves can be programmed using E- and L-. E- gives a distance along the drive line used for the taper. E0.2 will give a taper for the first/last 0.2 length units along the thread. L- is used to specify which ends of the thread get the taper. Program L0 for no taper (the default), L1 for entry taper, L2 for exit taper, or L3 for both entry and exit tapers.

The tool will pause briefly for synchronization before each threading pass, so a relief groove will be required at the entry unless the beginning of the thread is past the end of the material or an entry taper is used.

Unless using an exit taper, the exit move (traverse to original X) is not synchronized to the spindle speed. With a slow spindle, the exit move might take only a small fraction of a revolution. If the spindle speed is increased after several passes are complete, subsequent exit moves will require a larger portion of a revolution, resulting in a very heavy cut during the exit move. This can be avoided by providing a relief groove at the exit, or by not changing the spindle speed while threading.

The sample program g76.ngc shows the use of the G76 canned cycle, and can be previewed and executed on any machine using the sim/lathe.ini configuration.

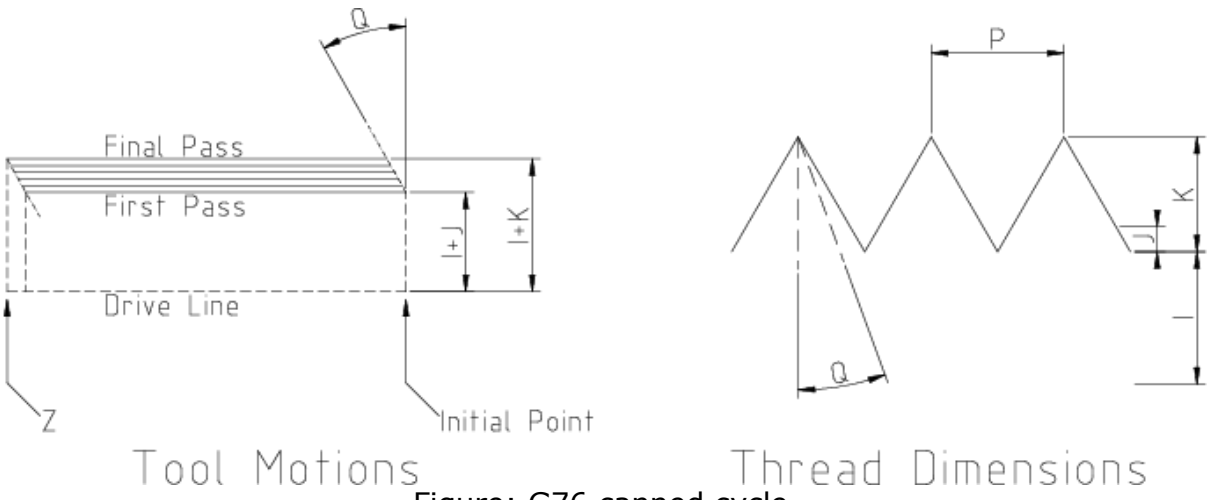


Figure: G76 canned cycle

This is how it works:

1) Before the start, the spindle rate is measured.
2) The feed for de z-axis is calculated: F = pitch * spindleRate
3) The CPU programmed such that a movement is started on the spindle pulse.
4) The movement is calculated and send to the CPU.
5) The movement is started when the spindle pulse passes.
6) Before the treading starts, the spindle-rate is measured, averaged and the feed is calculated from this.

Not that the inside and outside thread diameter are determined by the start position, the position before G76 and the I, K parameters.

3.6.19 Cancel Modal Motion - G80

Program G80 to ensure no axis motion will occur.

It is an error if:

- Axis words are programmed when G80 is active, unless a modal group 0 G code is programmed which uses axis words.

3.6.20 Canned Cycles - G81 to G89

The canned cycles G81 through G89 have been implemented as described in this section. Two examples are given with the description of G81 below.

All canned cycles are performed with respect to the currently selected plane. Any of the three planes (XY, YZ, and ZX) may be selected. Throughout this section, most of the descriptions assume the XY-plane has been selected. The behavior is always analogous if the YZ or XZ-plane is selected.

Rotational axis words are allowed in canned cycles, but it is better to omit them. If rotational axis words are used, the numbers must be the same as the current position numbers so that the rotational axes do not move.

All canned cycles use X, Y, R, and Z numbers in the NC code. These numbers are used to determine X, Y, R, and Z positions. The R (usually meaning retract) position is along the axis perpendicular to the currently selected plane (Z-axis for XY-plane, X-axis for YZ-plane, Y-axis for XZ-plane). Some canned cycles use additional arguments.

For canned cycles, we will call a number "sticky" if, when the same cycle is used on several lines of code in a row, the number must be used the first time, but is optional on the rest of the lines. Sticky numbers keep their value on the rest of the lines if they are not explicitly programmed to be different. The R number is always sticky.

In incremental distance mode: when the XY-plane is selected, X, Y, and R numbers are treated as increments to the current position and Z as an increment from the Z-axis position before the move involving Z takes place; when the YZ or XZ-plane is selected, treatment of the axis words is analogous. In absolute distance mode, the X, Y, R, and Z numbers are absolute positions in the current coordinate system.

The L number is optional and represents the number of repeats. L=0 is not allowed. If the repeat feature is used, it is normally used in incremental distance mode, so that the same sequence of motions is repeated in several equally spaced places along a straight line. In absolute distance mode, L > 1 means "do the same cycle in the same place several times," Omitting the L word is equivalent to specifying L=1. The L number is not sticky.

When L>1 in incremental mode with the XY-plane selected, the X and Y positions are determined by adding the given X and Y numbers either to the current X and Y positions (on the first go-around) or to the X and Y positions at the end of the previous go-around (on the repetitions). The R and Z positions do not change during the repeats.

The height of the retract move at the end of each repeat (called "clear Z" in the descriptions below) is determined by the setting of the retract mode: either to the original Z position (if that is above the R position and the retract mode is G98, OLD_Z), or otherwise to the R position. See Section 3.6.20

It is an error if:
- X, Y, and Z words are all missing during a canned cycle,
- a P number is required and a negative P number is used,
- an L number is used that does not evaluate to a positive integer,
- rotational axis motion is used during a canned cycle,
- inverse time feed rate is active during a canned cycle,
- cutter radius compensation is active during a canned cycle.

When the XY plane is active, the Z number is sticky, and it is an error if:
- the Z number is missing and the same canned cycle was not already active,
- the R number is less than the Z number.

When the XZ plane is active, the Y number is sticky, and it is an error if:
- the Y number is missing and the same canned cycle was not already active,
- the R number is less than the Y number.

When the YZ plane is active, the X number is sticky, and it is an error if:
- the X number is missing and the same canned cycle was not already active,
- the R number is less than the X number.

3.6.20.1 PRELIMINARY AND IN-BETWEEN MOTION

At the very beginning of the execution of any of the canned cycles, with the XY-plane

selected, if the current Z position is below the R position, the Z-axis is traversed to the R position. This happens only once, regardless of the value of L.

In addition, at the beginning of the first cycle and each repeat, the following one or two moves are made:

1. a straight traverse parallel to the XY-plane to the given XY-position,
2. a straight traverse of the Z-axis only to the R position, if it is not already at the R position.

If the XZ or YZ plane is active, the preliminary and in-between motions are analogous.

3.6.20.2 G81 CYCLE

The G81 cycle is intended for drilling. Program **G81 X... Y... Z... A... B... C... R... L...**

1. Preliminary motion, as described above.
2. Move the Z-axis only at the current feed rate to the Z position.
3. Retract the Z-axis at traverse rate to clear Z.

Example: Suppose the current position is (1, 2, and 3) and the XY-plane has been selected, and the following line of NC code is interpreted.

G90 G81 G98 X4 Y5 Z1.5 R2.8

This calls for absolute distance mode (G90) and OLD_Z retract mode (G98) and calls for the G81 drilling cycle to be performed once. The X number and X position are 4. The Y number and Y position are 5. The Z number and Z position are 1.5. The R number and clear Z are 2.8. Old Z is 3. The following moves take place.

1. a traverse parallel to the XY-plane to (4,5,3)
2. a traverse parallel to the Z-axis to (4,5,2.8)
3. a feed parallel to the Z-axis to (4,5,1.5)
4. a traverse parallel to the Z-axis to (4,5,3)

Example: Suppose the current position is (1, 2, and 3) and the XY-plane has been selected, and the following line of NC code is interpreted.

G91 G81 G98 X4 Y5 Z-0.6 R1.8 L3

This calls for incremental distance mode (G91) and OLD_Z retract mode (G98) and calls for the G81 drilling cycle to be repeated three times. The X number is 4, the Y number is 5, the Z number is -0.6 and the R number is 1.8. The initial X position is 5 (=1+4), the initial Y position is 7 (=2+5), the clear Z position is 4.8 (=1.8+3), and the Z position is 4.2 (=4.8-0.6). Old Z is 3.
The first move is a traverse along the Z-axis to (1,2,4.8), since old Z < clear Z.
The first repeat consists of 3 moves.

1. a traverse parallel to the XY-plane to (5,7,4.8)
2. a feed parallel to the Z-axis to (5,7, 4.2)
3. a traverse parallel to the Z-axis to (5,7,4.8)

The second repeat consists of 3 moves. The X position is reset to 9 (=5+4) and the Y position to 12 (=7+5).

1. a traverse parallel to the XY-plane to (9,12,4.8)
2. a feed parallel to the Z-axis to (9,12, 4.2)
3. a traverse parallel to the Z-axis to (9,12,4.8)

The third repeat consists of 3 moves. The X position is reset to 13 (=9+4) and the Y position to 17 (=12+5).

1. a traverse parallel to the XY-plane to (13,17,4.8)
2. a feed parallel to the Z-axis to (13,17, 4.2)
3. a traverse parallel to the Z-axis to (13,17,4.8)

3.6.20.3 G82 CYCLE

The G82 cycle is intended for drilling. Program **G82 X... Y... Z... A... R... L... P...**

1. Preliminary motion, as described above.
2. Move the Z-axis only at the current feed rate to the Z position.
3. Dwell for the P number of seconds.
4. Retract the Z-axis at traverse rate to clear Z.

3.6.20.4 G83 CYCLE

The G83 cycle (often called peck drilling) is intended for deep drilling or milling with chip breaking. The retracts in this cycle clear the hole of chips and cut off any long stringers (which are common when drilling in aluminum). This cycle takes a Q number which represents a "delta" increment along the Z-axis**.**

Program G83 X... Y... Z... A... R... L... Q...

1. Preliminary motion, as described above.
2. Move the Z-axis only at the current feed rate downward by delta or to the Z position, whichever is less deep.
3. Rapid back out to the clear_z.
4. Rapid back down to the current hole bottom, backed off a bit.
5. Repeat steps 1, 2, and 3 until the Z position is reached at step 1.
6. Retract the Z-axis at traverse rate to clear Z.

It is an error if:

• the Q number is negative or zero.

3.6.20.5 G73 CYCLE

The G73 cycle (often called peck drilling) is intended for deep drilling or milling with chip breaking. The retracts in this cycle clear the hole of chips and cut off any long stringers (which are common when drilling in aluminum). This cycle takes a Q number which represents a "delta" increment along the Z-axis**.**

Program G73 X... Y... Z... A... R... L... Q...

1. Preliminary motion, as described above.
2. Move the Z-axis only at the current feed rate downward by delta or to the Z position, whichever is less deep.
3. Rapid back out but only with increment Q, this is the difference with G83 above.
4. Rapid back down to the current hole bottom, backed off a bit.
5. Repeat steps 1, 2, and 3 until the Z position is reached at step 1.
6. Retract the Z-axis at traverse rate to clear Z.

It is an error if:
• the Q number is negative or zero.

3.6.20.6 G84 CYCLE

The G84 cycle is intended for right hand tapping.
Program **G84 X... Y... Z... A... B... C... R... L...**

1. Preliminary motion, as described above.
2. Move the Z-axis only at the current feed rate/per revolution to the Z position.
   So assume the spindle is running M3 S600. Then and F value of F1 will give A feed of 600 /minute. Feed starts synchronized with spindle pulse allowing to tap the same hole again.
3. When Z position reached, reverse spindle M4. (Waits until spindle ramp-up and new measurement of spindle speed)
4. Retract the Z-axis at the current feed rate to clear Z.

3.6.20.7 G74 CYCLE

The G74 cycle is intended for left hand tapping.
Program **G74 X... Y... Z... A... B... C... R... L...**

5. Preliminary motion, as described above.
6. Move the Z-axis only at the current feed rate/per revolution to the Z position.
   So assume the spindle is running M3 S600. Then and F value of F1 will give A feed of 600 /minute. Feed starts synchronized with spindle pulse allowing to tap the same hole again.
7. When Z position reached, reverse spindle M4. (Waits until spindle ramp-up and new measurement of spindle speed)
8. Retract the Z-axis at the current feed rate to clear Z.

3.6.20.8 G85 CYCLE

The G85 cycle is intended for boring or reaming, but could be used for drilling or milling. Program G85 X… Y… Z… A… B… C… R… L…
Preliminary motion, as described above.
Move the Z-axis only at the current feed rate to the Z position.
Retract the Z-axis at the current feed rate to clear Z.

3.6.20.9 G86 CYCLE

The G86 cycle is intended for boring. This cycle uses a P number for the number of seconds to dwell. Program G86 X… Y… Z… A… B… C… R… L… P…
Preliminary motion, as described above.
Move the Z-axis only at the current feed rate to the Z position.
Dwell for the P number of seconds.
Stop the spindle turning.
Retract the Z-axis at traverse rate to clear Z.
Restart the spindle in the direction it was going.

The spindle must be turning before this cycle is used. It is an error if:
the spindle is not turning before this cycle is executed.

3.6.20.10 G87 CYCLE

The G87 cycle is intended for back boring.
Program **G87 X... Y... Z... A... R... L... I... J... K...**

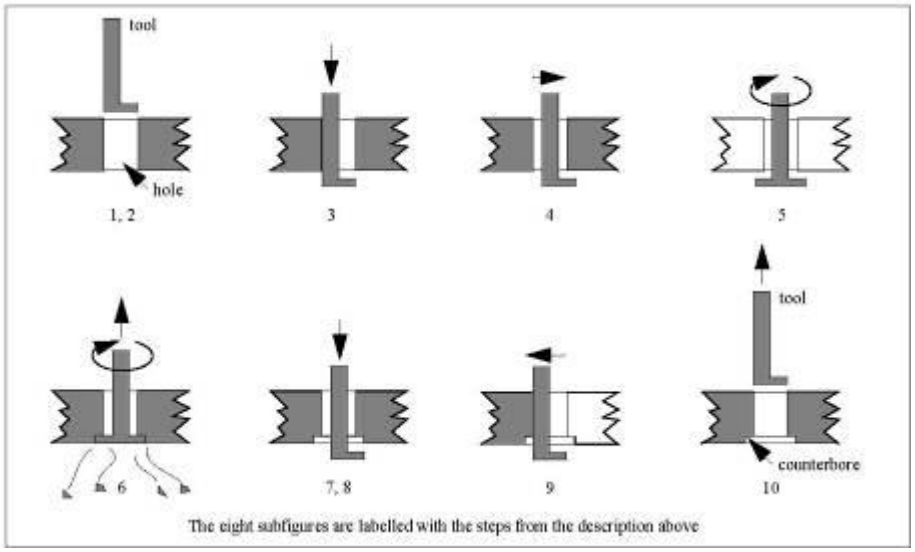
The situation, as shown in Figure 3-1, is that you have a through hole and you want to counter bore the bottom of hole. To do this you put an L-shaped tool in the spindle with a cutting surface on the UPPER side of its base. You stick it carefully through the hole when it is not spinning and is oriented so it fits through the hole, then you move it so the stem of the L is on the axis of the hole, start the spindle, and feed the tool upward to make the counter bore. Then you stop the tool, get it out of the hole, and restart it.

This cycle uses I and J numbers to indicate the position for inserting and removing the tool. I and J will always be increments from the X position and the Y position, regardless of the distance mode setting. This cycle also uses a K number to specify the position along the Z-axis of the controlled point top of the counter bore. The K number is a Z-value in the current coordinate system in absolute distance mode, and an increment (from the Z position) in incremental distance mode.

1. Preliminary motion, as described above.
2. Move at traverse rate parallel to the XY-plane to the point indicated by I and J.
3. Stop the spindle in a specific orientation.
4. Move the Z-axis only at traverse rate downward to the Z position.
5. Move at traverse rate parallel to the XY-plane to the X,Y location.
6. Start the spindle in the direction it was going before.
7. Move the Z-axis only at the given feed rate upward to the position indicated by K.
8. Move the Z-axis only at the given feed rate back down to the Z position.
9. Stop the spindle in the same orientation as before.
10. Move at traverse rate parallel to the XY-plane to the point indicated by I and J.
11. Move the Z-axis only at traverse rate to the clear Z.
12. Move at traverse rate parallel to the XY-plane to the specified X,Y location.
13. Restart the spindle in the direction it was going before.

When programming this cycle, the I and J numbers must be chosen so that when the tool is stopped in an oriented position, it will fit through the hole. Because different cutters are made differently, it may take some analysis and/or experimentation to determine appropriate values for I and J.

Figure 3-1 **G87 Cycle**



3.6.20.11 G88 CYCLE

The G88 cycle is intended for boring. This cycle uses a P word, where P specifies the
number of seconds to dwell. Program **G88 X... Y... Z... A... R... L... P...**
Preliminary motion, as described above.

1. Move the Z-axis only at the current feed rate to the Z position.
2. Dwell for the P number of seconds.
3. Stop the spindle turning.
4. Stop the program so the operator can retract the spindle manually.
5. Restart the spindle in the direction it was going.

3.6.20.12 G89 CYCLE

The G89 cycle is intended for boring. This cycle uses a P number, where P specifies the
number of seconds to dwell. program **G89 X... Y... Z... A... R... L... P...**

1. Preliminary motion, as described above.
2. Move the Z-axis only at the current feed rate to the Z position.
3. Dwell for the P number of seconds.
4. Retract the Z-axis at the current feed rate to clear Z.

3.6.21 Set Distance Mode - G90 and G91

To make the current point have the coordinates you want (without motion), program G92 X… Interpretation of RS274/NGC code can be in one of two distance modes: absolute or incremental.

To go into absolute distance mode, program G90. In absolute distance mode, axis numbers (X, Y, Z, A, B, C) usually represent positions in terms of the currently active coordinate system.

To go into incremental distance mode, program G91. In incremental distance mode, axis numbers (X, Y, Z, A, B, C) usually represent a distance from the current values of the numbers.

I and J numbers always represent increments, regardless of the distance mode

3.6.22 **Coordinate System Offsets - G92, G92.1, G92.2, G92.3**

To make the current point have the coordinates you want (without motion), program
G92 X... Y... Z... A... , where the axis words contain the axis numbers you want. All axis words are optional, except that at least one must be used. If an axis word is not used for a given axis, the coordinate on that axis of the current point is not changed.

It is an error if:
- all axis words are omitted.

When G92 is executed, the origin of the currently active coordinate system moves. To do this, origin offsets are calculated so that the coordinates of the current point with respect to the moved origin are as specified on the line containing the G92. In addition, parameters 5211 to 5216 are set to the X, Y, Z, A, B, and C-axis offsets. The offset for an axis is the amount the origin must be moved so that the coordinate of the controlled point on the axis has the specified value.

Here is an example. Suppose the current point is at X=4 in the currently specified coordinate system and the current X-axis offset is zero, then G92 x7 sets the X-axis offset to -3, sets parameter 5211 to -3, and causes the X-coordinate of the current point to be 7.

The axis offsets are always used when motion is specified in absolute distance mode using any of the nine coordinate systems (those designated by G54 - G59.3). Thus all nine coordinate systems are affected by G92.

Being in incremental distance mode has no effect on the action of G92.

Non-zero offsets may already be in effect when the G92 is called. If this is the case, the new value of each offset is A+B, where A is what the offset would be if the old offset were zero, and B is the old offset. For example, after the previous example, the X-value of the current point is 7. If G92 x9 is then programmed, the new X-axis offset is -5, which is calculated by [[7-9] + -3].

To reset axis offsets to zero, program **G92.1** or **G92.2**. G92.1 sets parameters 5211 to 5216 to zero, whereas G92.2 leaves their current values alone.
To set the axis offset values to the values given in parameters 5211 to 5216, program **G92.3**.

You can set axis offsets in one program and use the same offsets in another program. Program G92 in the first program. This will set parameters 5211 to 5216. Do not use G92.1 in the remainder of the first program. The parameter values will be saved when the first program exits and restored when the second one starts up. Use G92.3 near the beginning of the second program. That will restore the offsets saved in the first program. If other programs are to run between the program that sets the offsets and the one that restores them,

make a copy of the parameter file written by the first program and use it as the parameter file for the second program.

3.6.23 Set Feed Rate Mode - G93, G94, G95

Three feed rate modes are recognized, depending on selected mode the Feed of the axes is calculated differently:

- **G93** inverse time, a move is completed in **1/F minutes**. For example if F=6 the move is completed in 10 seconds. When G93 is active, the F must be specified on every line containing G1,G3 or G3.
- **G94** units per minute, this is the normal mode for milling**, F means units per minute**, in millimeter mode mm/minute, in INCH mode inch/minute.
- **G95 Units per revolution,** here the F word is the number of units that should be cut per spindle revolution. So the feed of the axes is depending on the rotation speed of the spindle. G95 F2 means cut 2mm every spindle revolution, so when S=500, the feed for XZ would be FEED = F * S = 2 * 500 = 1000.

3.6.24 Spindle Control Mode – G96, G97

Two spindle control modes are possible, depending on the mode the spindle speed is calculated differently:

- **G96 (Only for Lathe)**, select constant surface speed, S is now specified as meters per minute in mm mode (G21) or Feet per minute in inch mode (G20). This means that the spindle speed is adapted automatically when the Radius changes. Suppose you program G96 S150 in millimeter mode, the spindle speed is calculated by: **RPM = S / (2 * PI * X)**, X is the radius. So your X zero must be where the diameter is zero.
  Example: Your actual X position is 100 (100 millimeter = 0.1 meter), you program G96 S150 D1000, this would result in a spindle RPM of 150 / (2 * PI * 0.1) = 238.7 rev/min. D1000 limits the maximum speed to 1000 RPM.
- **G97** is normal RPM mode, S specifies the RPM.

3.6.25 Set Canned Cycle Return Level - G98 and G99

When the spindle retracts during canned cycles, there is a choice of how far it retracts:
(1) retract perpendicular to the selected plane to the position indicated by the R word, or
(2) retract perpendicular to the selected plane to the position that axis was in just before the canned cycle started (unless that position is lower than the position indicated by the R word, in which case use the R word position).

To use option (1), program G99. To use option (2), program G98. Remember that the R word has different meanings in absolute distance mode and incremental distance mode.

3.7 INPUT M CODES

M codes of the RS274/NGC language are shown in Table 3-6

Table 3-6 **M Codes**

M Code	Meaning
M0	program stop
M1	optional program stop
M2	program end
M3	turn spindle clockwise
M4	turn spindle counterclockwise
M5	stop spindle turning
M6	tool change
M7	mist coolant on
M8	flood coolant on
M9	mist and flood coolant off
M20	Plasma Torch Height Control ON
M21	Plasma Torch Height Control OFF
M22	M23 Q.. Set Plasma THC set point value;
M30	program end, spindle and coolants off and rewind.
M48	enable speed and feed overrides
M49	disable speed and feed overrides
M60	program stop, use this with nesting instead of M60, so that the spindle/coolants remain on during transition from one to the next run.
M54	set general purpose output for CPU5B
M55	clear general purpose output for CPU5B
M56	read general purpose input for CPU5B
M80	Drive enable ON
M81	Drive enable OFF
M90	Standard Head/spindle
M91	Alternate Head/2nd spindle
M93	Alternate Head/3rd spindle
M95	Alternate Head/Probe
M97	Alternate Head/Camera
	Note that a head may as well be e.g. a tangential knife configuration
	Specials for 3D printing
M104	M104 S.. Set extruder temperature, (M104 S50, sets temperature to 50 degree Celsius).
M106	M106 S.. Work piece cooling FAN ON optionally with S=0-255, for 0-100% PWM.
M107	M107 Work piece FAN off.
M109	M109 S.. Set extruder temperature and wait until reached.
M143	M143 S.. Maximum Hot-end temperature to prevent overheating.
M140	M140 S.. Bed temperature.

M190	M190 Wait for bed temperature reached target.

3.7.1 **Program Stopping and Ending - M0, M1, M2, M30, M60**

To halt a running program temporarily, program **M0**. If a program is stopped by an **M0**, pressing the cycle start button will restart the program at the following line, so the program will continue.

To optionally halt a program when the stopM1 check in the user interface is checked program **M1**

program **M30** for next effects:

- Selected plane is set to CANON_PLANE_XY (like G17).
- Distance mode is set to MODE_ABSOLUTE (like G90).
- Feed rate mode is set to UNITS_PER_MINUTE (like G94).
- Feed and speed overrides are set to ON (like M48).
- Cutter compensation is turned off (like G40).
- The spindle is stopped (like M5).
- The current motion mode is set to G_1 (like G1).
- Coolants are turned off (like M9).
- Note that the coordinate system are no longer reset, I modified this behavior because I have broken a lot of bits due to this so I modified it.
- Program is re-winded to the first line, ready for next start.
- All 3D printer heating OFF.

Program **M60** instead of M30 if the spindle and coolants should remain ON, this is usefully with nesting.

3.7.2 **Spindle/Head Control - M3, M4, M5, M90-M97**

To start the spindle turning clockwise at the currently programmed speed, program **M3**.
To start the spindle turning counterclockwise at the currently programmed speed, program **M4**.
To stop the spindle from turning, program **M5**.
It is OK to use M3 or M4 if the spindle speed is set to zero. If this is done (or if the speed override switch is enabled and set to zero), the spindle will not start turning. If, later, the spindle speed is set above zero (or the override switch is

turned up), the spindle will start turning. It is OK to use M3 or M4 when the spindle is already turning or to use M5 when the spindle is already stopped.

If there are more than one spindles in your machine, or of you have a mounted touch probe on th Z slide or a camera you can select the offset/IO you want

- M90 standard spindle
- M91 alternate 2nd spindle
- M92 alternate 3rd spindle
- M95 touch probe
- M97 Camera

Each spindle is connected to different set of outputs and has different parameters.
Alternate spindle 1 and 2 may have defined offsets for x,y,z.
Because this option is rarely used by customers, you have to perform the settings yourself by editing the cnc.ini file.

There are 5 sets of parameters for each spindle configuration. For the 2nd 3rd 4th and 5th spindle configuration you can set the axis offsets with respect to the 1nd spindle. Note that the 4th and 4th spindle configuration is used for a touch probe and camera.
Example system:
1. Spindle, 2. tan knife, and possible 3. oscillating tan knife, 4. Touch probe, 5 Camera.

Because this function is used only by a few customers, it is left out of the setup in the GUI. The settings have to be added manually in the cnc.ini settings file:

```
[SPINDLE_0]
spindleRampUpTime = 1.00
spindleNmax = 24000.00
spindleNmin = 1000.00
spindleUseRPMSensor = 0
onOffOutputPortID = 0 ;0: Standard tool output, 1-9: AUX1-AUX9
directionOutputPortID = 0 ;0: Standard tool output, 1-9: AUX1-AUX9, -2: MIST COOLANT Output
pwmOutputPortID = 1 ;0: Standard PWM output, 1-3: PWM1-PWM3 Output

[SPINDLE_1]
spindleRampUpTime = 1.00
spindleNmax = 24000.00
spindleNmin = 1000.00
spindleUseRPMSensor = 0
onOffOutputPortID = 0 ;0: Standard tool output, 1-9: AUX1-AUX9
directionOutputPortID = 0 ;0: Standard tool output, 1-9: AUX1-AUX9, -2: MIST COOLANT Output
pwmOutputPortID = 1 ;0: Standard PWM output, 1-3: PWM1-PWM3 Output
xOffset = 0.0000
yOffset = 0.0000
zOffset = 0.0000

[SPINDLE_2]
spindleRampUpTime = 1.00
spindleNmax = 24000.00
spindleNmin = 1000.00
spindleUseRPMSensor = 0
onOffOutputPortID = 0 ;0: Standard tool output, 1-9: AUX1-AUX9
directionOutputPortID = 0 ;0: Standard tool output, 1-9: AUX1-AUX9, -2: MIST COOLANT Output
pwmOutputPortID = 1 ;0: Standard PWM output, 1-3: PWM1-PWM3 Output
```

```
xOffset = 0.0000
yOffset = 0.0000
zOffset = 0.0000

[SPINDLE_3]
;Mounted Probe M95
xOffset = 0.0000
yOffset = 0.0000
zOffset = 0.0000
onOffOutputPortID = 0 ;0: Standard tool output, 1-9: AUX1-AUX9

[SPINDLE_4]
;Mounted camera M97
xOffset = 0.0000
yOffset = 0.0000
zOffset = 0.0000
onOffOutputPortID = 0 ;0: Standard tool output, 1-9: AUX1-AUX9
```

The x,y,z Offset parameter for the [SPINDLE_1] and [SPINDLE_2] [SPINDLE_3] and [SPINDLE_4] configuration are offsets with respect to [SPINDLE_0]. Every spindle has its own parameters including IO ports for switching on/off and controlling the speed.

3.7.3 **Tool Change - M6**

To change a tool in the spindle from the tool currently in the spindle to the tool most
recently selected (using a T word - see Section 3.7.3), program **M6**. When the tool change is complete:

- The spindle will be stopped.
- The tool that was selected (by a T word on the same line or on any line after the previous tool change) will be in the spindle. The T number is an integer giving the changer slot of the tool (not its id).
- If the selected tool was not in the spindle before the tool change, the tool that was in the spindle (if there was one) will be in its changer slot.
- The coordinate axes will be stopped in the same absolute position they were in before the tool change (but the spindle may be re-oriented).
- No other changes will be made. For example, coolant will continue to flow during the tool change unless it has been turned off by an M9.

The tool change may include axis motion while it is in progress. It is OK (but not useful) to program a change to the tool already in the spindle. It is OK if there is no tool in the selected slot; in that case, the spindle will be empty after the tool change. If slot zero was last selected, there will definitely be no tool in the spindle after a tool change.

The tool change command will call the **change_tool** subroutine inside macro.cnc.
You can adapt the behavior for your own needs in this function e.g:

- Perform automatic tool-length measurement
- Perform tool change with an automatic tool changer.

For a (nonfunctional) example of how to implement automatic tool change for a 16-tool changer. see the contents of the default_macro.cnc file at the end of this document. It checks whether current tool is already in the spindle. It check

that the tool number is in range of 1-4. Then it first drops current tool and picks the new tool:

3.7.4 **Coolant Control - M7, M8, M9**

To turn mist coolant on, program M7. To turn flood coolant on, program M8. To turn all coolant off, program M9. It is always OK to use any of these commands, regardless of what coolant is on or off.

3.7.5 Feed-Speed Override Control - M48-M53

M48 Enable feed and speed override
M49 Disable feed and speed override
M50 P.. Set feed Override to given P value, if P value is less than zero feed override is disabled and the value remains as is.
M51 P.. Set speed Override to given P value, if P value is less than zero speed override is switched off.
M52 <P1>Enable feed Override by analog input. M52 P0 disable feed Override by analog input. P1 is optional.
M53 <P1> Enable feed Hold input. M53 P0 disable feed Hold input. P1 is optional.

To enable the speed and feed override switches, program M48. To disable both switches, program M49.

3.7.6 IO M Functions

3.7.7 Standard CNC IO - M3..M9, M80..M87

To control the outputs, these functions have been added besides the standard M-Functions.

Standard, according to [NIST]
M3 PWM according S value, TOOLDIR = on
M4 PWM according S value, TOOLDIR = off
M5 PWM off, TOOLDIR off.
M7 Mist on
M8 Flood on
M9 Mist/Flood off

Additional, to support the features of the EDINGCNC CPU's
M80 drive enable on
M81 drive enable off
~~M84 TOOLDIR on~~ (no longer supported)
~~M85 TOOLDIR off~~ (No longer supported)
~~M86 PWM according S value~~ (No longer supported, use M54 instead)
~~M87 PWM off~~ (No longer supported, used M54 instead)

3.7.8 General purpose IO of CPU5B - M54, M55 and M56

M54 Px
Set output x.
M54 P1 (set AUX1 out to 1)

M54 Ex Qy
Set PWM output x to promille value y (0 <= y <= 1000)
M54 E2 Q500 (Set PWM2 to 50% PWM)

M55 Px

Clear output x.

M55 P1 (set AUX1 out to 0)

M56 Px
Read input x. result stored on #5399

```
M56 P3 (Read AUX in 3)
If [#5399 == 1]
   Msg "AUX3=ON"
Else
   Msg "AUX3=OFF"
endif
```

M56 Px Ly Qy
Read digital input and specify wait mode, result stored in #5399
Px: x is input number
L0: do not wait
L1: Wait for High
L2: Wait for Low
Qy: y is timeout

```
M56 P3 L2 Q30 (Read AUX in 3)
If [#5399 == -1]
   Errmsg "Timeout while waiting for AUX3 becoming low"
Else
   Msg "AUX3 is off"
Endif
```

Note that we use wait (L2) here, in case if time-out, the value of #5399 is -1.

M56 Ex
Read analogue input, result stored in #5399
Ex: x is input number

M56 E3
Msg "analog value is "#5399

Read other inputs using M56

M56 Px

Home Inputs: x = 51 – 56 (X .. C)
Probe Input: x = 61
Sync Input: x = 62
HWA Input: x= 63
HWB Input: x= 64
ESTOP1: x=65
ESTOP2: x=66
EXTERR: x=67

Example, read home-input of X axis:

```
M56 P51
If [#5399 == 1]
   Msg "HOMEX=ON"
Else
   Msg "HOMEX=OFF"
endif
```

Optional IOCARD

If the new I2C GPIO CARD is used, the IO number P or E is specified as follows:
103 means card 1 port 3.
208 means card 2 port 8

3.7.9 A axis clamping M26, M27

The A axis is often used to rotate the work piece on the machine and then do milling on that side of the work piece. To be sure the work piece is fixed at its place an axis clamp (Brake) can be used for A.

M26 P1 enables the clamp and the optional P1 specifies the output that controls the brake, P1 means AUX 1. (See also M54)

M27 P1 disables the clamp and the optional P1 here specifies the output controlling the brake.

When M26 is active, the software does not accept movements g-code and jogging, an error message will appear that the A axis is clamped.

3.7.10 **Torch height control M20, M21**

M20 THC off
M21 THC on

3.7.11 **M Functions for 3D printing**

M1 :All heating and Fans off
M104 S :Set extruder temperature, (M104 S50, sets temperature to 50 degree Celsius).
M106 S.. : Work piece cooling FAN ON optionally with S=0-100, for 0-100% PWM
M107 :Work piece FAN off .
M109 S.. :Set extruder temperature and wait until reached.
M143 S.. :Maximum Hot-end temperature to prevent overheating.
M140 S.. :Bed temperature
M143 S.. :Set max extruder temperature
M190 S.. :Set Bed temperature and wait until reached.
M100 P.. :Change a axis resolution on the fly by a factor (factor 1 is standard, 1.01 is 1% more, 0.99 is 1% less.

3.7.12 **M Function override and user m-functions**

The system allows M functions in the range of M1..M999.
This means there are many un-used m-functions.
The user can create his own M-Function by creating a subroutine for it in the macro.cnc. or usermacro.cnc file
e.g:

```
sub m100
   ;Do your stuff here
   Msg "my M100"
   M54 p1; switch on AUX output 1
end sub
```

So if the g-code file performs M100, the subroutine will be called.
Also parameters in the form M100 S100 is possible.
In the subroutine the parameter can be accessed using #19 (#1 .. #26 accesses parameter A..Z), S is number 19 in the alphabet.

This creates further possibilities:
It is possible to override existing M functions as well.
Suppose you want additional function for M3 which is spindle on standard.
If you spindle has an output speed-reached, you can wait for it like this:

```
sub m3
  msg "my customized m3"
  m3 s#19;The real m3, inside sub routine this m3 will execute the real m3
  m56 P1 L2 Q60 ; Wait max. 1 minute for input 1 to become low
endsub
```

When you are inside your m3 subroutine and perform an m3, then the subroutine is not called, but the default m3 is called in stead.

There are some limitations on using the M Function override:

- There must be nothing else on the G-code line as the M-Function, e.g.
  N2000 M3 S1000 is OK.
  N2000 M3 M8 G1X100 is NOT OK!

- The user parameters 1-26 will be used as parameter and will be overwritten when M-Function subroutines are used. Take this into account and do not use #1..#26 in your program if you use this functionality. Variable 1-26 match with the letter of the alphabet, #1 will get the value of parameter A, #26 will get the value of parameter Z.

3.8 OTHER INPUT CODES

3.8.1 Set Feed Rate - F

To set the feed rate, program F… . The application of the feed rate is as described in
Section 2.1.2.5, unless inverse time feed rate mode is in effect, in which case the feed rate is as described in Section 3.6.23.

3.8.2 Set Spindle Speed - S

To set the speed in revolutions per minute (rpm) of the spindle, program S… . The spindle will turn at that speed when it has been programmed to start turning. It is OK to program an S word whether the spindle is turning or not. If the speed override switch is
enabled and not set at 100%, the speed will be different from what is programmed. It is OK to program S0; the spindle will not turn if that is done. The CPU's that support PWM output will have its PWM value set conform the requested spindle speed if the spindle is turned on.

It is an error if:

- the S number is negative.

3.8.3 Select Tool - T

To select a tool, program T…, where the T number is the carousel slot for the tool. The tool is not changed until an M6 is programmed (see Section 3.7.3). The T word may appear on the same line as the M6 or on a previous line. It is OK, but not normally useful, if T words appear on two or more lines with no tool change. The carousel may move a lot, but only the most recent T word will take effect at the next tool change. It is OK to
program T0; no tool will be selected. This is useful if you want the spindle to be empty after a tool change.

It is an error if:

- a negative T number is used,
- a T number larger than the number of slots in the carousel is used.

On some machines, the carousel will move when a T word is programmed, at the same time machining is occurring. On such machines, programming the T word several lines before a tool change will save time. A common programming practice for such machines is to put the T word for the next tool to be used on the line after a tool change. This maximizes the time available for the carousel to move.

3.9 ORDER OF EXECUTION

The order of execution of items on a line is critical to safe and effective machine operation. Items are executed in the order shown in Table 3-7 if they occur on the same line.

Table 3-7 Order of execution

1. comment (includes message).
2. set feed rate mode (G93, G94 -inverse time or per minute).
3. set feed rate (F).
4. set spindle speed (S).
5. select tool (T).
6. change tool (M6).
7. spindle on or off (M3, M4, M5).
8. coolant on or off (M7, M8, M9).
9. enable or disable overrides (M48, M49).
10. dwell (G4).
11. set active plane (G17, G18, G19).
12. set length units (G20, G21).
13. cutter radius compensation on or off (G40, G41, G42)
14. cutter length compensation on or off (G43, G49)
15. coordinate system selection (G54, G55, G56, G57, G58, G59, G59.1, G59.2, G59.3).
16. set path control mode (G61, G61.1, G64)
17. set distance mode (G90, G91).
18. set retract mode (G98, G99).
19. home (G28, G30) or change coordinate system data (G10) or set axis offsets (G92, G92.1, G92.2, G94).
20. perform motion (G0 to G3, G80 to G89), as modified (possibly) by G53.
21. stop(M0,M1, M2,M30,M60).

4 Language extensions

To provide additional flexibility, I created some extensions in the language that allow for programming.

4.1 FLOW CONTROL

You can use the following flow control commands in a job:
IF[x]-ELSE-ENDIF constructs to define x dependent execution
WHILE[x]-ENDWHILE constructs to define x dependent repeated execution
SUB <name>-ENDSUB constructs to define a subroutine
GOSUB <name> construct to call a subroutine

4.2 SUPPORTED OPERATIONS ON EXPRESSIONS

4.2.1 unary operations

abs	absolute value
acos	arc cosine
asin	arc sine
atan[y]/[x]	arc tangent
cos	cosine
exp	e raised to
fix	round down
fup	round
int	integer part
ln	natural log of
round	round
sin	sine
sqrt	square root
tan	tangent
not	logical not

4.2.2 binary operations:

/	divided by
mod	modulo
**	power
*	times
and	logic and
xor	logic exclusive or
-	minus
or	logic nonexclusive or
+	plus
>	greater then
>=	greater than or equal
<	less then
<=	less than or equal
==	is equal
<>	not equal
band	bitwise and
bxor	bitwise exclusive or
bor	bitwise nonexclusive or
<<	shift left
>>	shift right

See also B.2 for examples on expressions.

4.2.3 An example:

```
sub do_circle_holes

#1=0
g0 z1 x0 y0
while [#1 <> 360]
   #2 = [10 * sin[#1]]
   #3 = [10 * cos[#1]]
   g0 x[#3] y[#2]
   g1 z-1
   g1 z1
  #1 = [#1 + 30]

  if [#1 == 360]
     msg "Done"
  else
     msg "processing at angle "#1
  endif

endwhile

endsub

gosub do_circle_holes
m30
```

This example drills holes at a circle with a radius of 10, each 30 degrees. The code that performs this is put in a subroutine, which can be called as many times as needed in the main program.

4.2.4 Special interpreter commands, non G-Code

Messages:

```
Msg "Hello there, the value of #1 = "#1" and the value of #2 = "#2
```

4.2.4.1 ERRMSG

Same as Msg, but this one generates an error

4.2.4.2 STORE POSITION

```
SP <filename> [0 or 1]
```

This command stores the actual position in given file name.
The extra parameter 0 means create the file.
1 means add to existing file.
If only file name is given, the position is added to existing file.

4.2.4.3 DLGMSG

Gives a dialog message for an interactive g-code program.

DlgMsg <dialog message> <par1Name> <par1ParNumber> ... <par12Name> <par12ParNumber>

Example:
DlgMsg "Give parameters" par1 100 par2 101
The dialog woll have an OK and a Cancel button.
When the user selects OK, variable #5398 is set to 1 and the program automatically continues. When the user selects CANCEL, variable #5398 is set to -1, program continues. Just try and you will see what this is about.
if <dialog message>.png picture exist it will be show.

During render mode, when the program is loaded and parsed to show in the graphics the dialogs will not appear logically, in that case #5398 is set to 1 indicating OK.
So you must give your variables a good predefined value that works.
You can also use "if [#5397 == 0]" this indicates normal running mode.
#5397 = 1 during the render mode.

EXAMPLE of dlg msg:
For this example we created a subroutine associated with user_3 button in the UI.

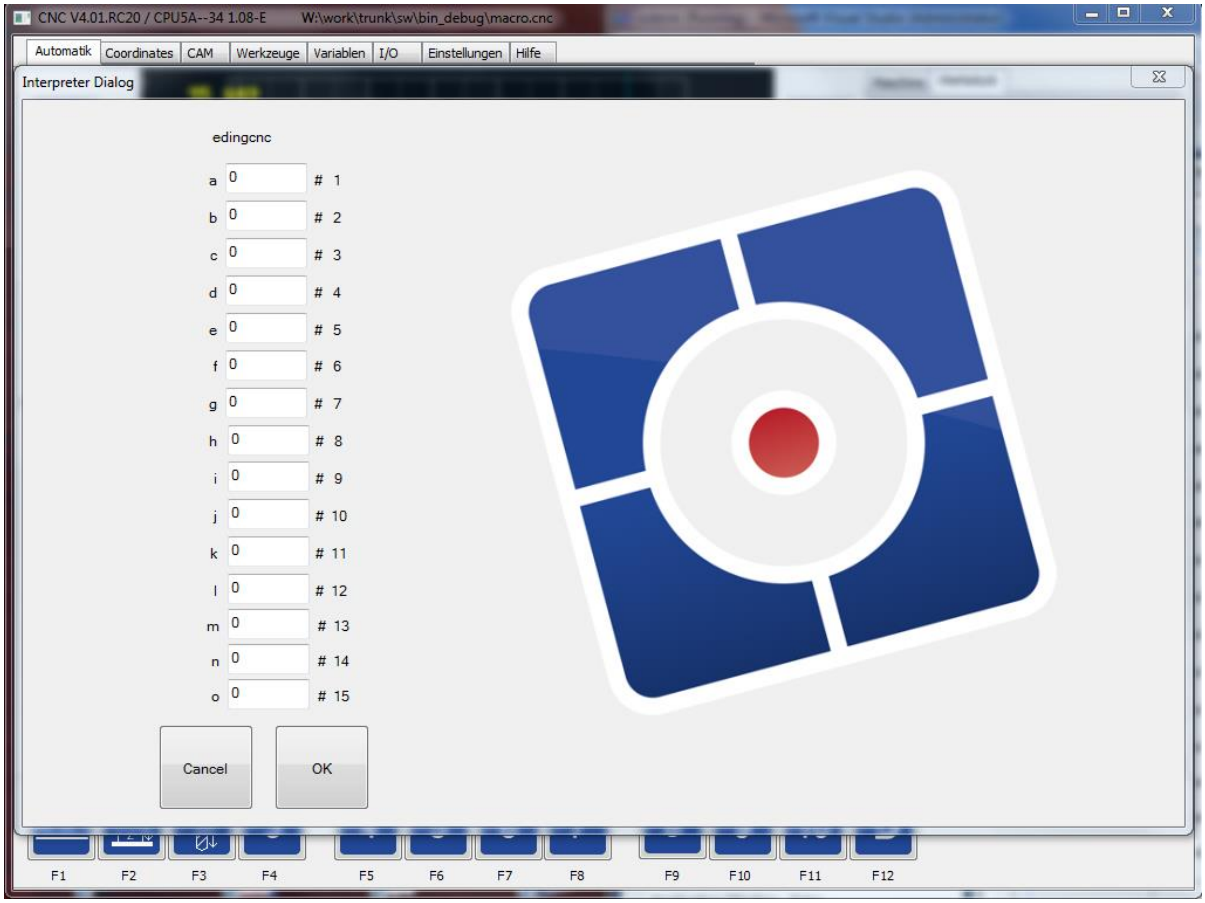
```
Sub user 3 ;Example of dlgmsg
    #1 = 0
    #2 = 0
    #3 = 0
    #4 = 0
    #5 = 0
    #6 = 0
    #7 = 0
    #8 = 0
    #9 = 0
    #10 = 0
    #11 = 0
    #12 = 0

    ;dlgmsg will pop up a dialog with picture EdingCNC.png from c:\program files
(x86)\cnc4.01\dialogPictures directory
    dlgmsg "edingcnc" "A" 1 "B" 2 "C" 3 "D" 4 "E" 5 "F" 6 "G" 7 "H" 8 "I" 9 "J" 10 "K" 11 "L"
12

    if [#5398 == 1]
        msg "OK #1="#1 "#2="#2 "#3="#3 "#4="#4 "#5="#5 "#6="#6 "#7="#7 "#8="#8 "#9="#9
"#10="#10 "#11="#11 "#12="#12
    else
        msg "CANCEL #1="#1 "#2="#2 "#3="#3 "#4="#4 "#5="#5 "#6="#6 "#7="#7 "#8="#8 "#9="#9
"#10="#10 "#11="#11 "#12="#12
    endif

Endsub
```

The dialog looks like this:


CNC V4.01.RC20 / CPU5A--34 1.08-E W:\work\trunk\sw\bin_debug\macro.cnc
Automatik
Coordinates
CAM
Werkzeuge
Variablen
I/O
Einstellungen
Hilfe
Interpreter Dialog
edingcnc
a 0 # 1
b 0 # 2
c 0 # 3
d 0 # 4
e 0 # 5
f 0 # 6
g 0 # 7
h 0 # 8
i 0 # 9
j 0 # 10
k 0 # 11
l 0 # 12
m 0 # 13
n 0 # 14
o 0 # 15
Cancel
OK
F1
F2
F3
F4
F5
F6
F7
F8
F9
F10
F11
F12

4.2.4.4 LogFile, LogMsg

Log anything to a file, first create a new file or open the file using LogFile, then write to the file using LogMsg.

```
LogFile <fileName> <1=append, 0=open new>
LogMsg  your message
```

Example:
LogFile "text.txt" 1
LogMsg "Hi, the current position of X is "#5001

Now check the contents of file text.txt.

4.2.4.5 TCAGuard [on | off]

Switches on or off the tool change area guard.
This is used during the rendering process, where the job file is checked for collisions with the machine area and tool change area.

4.2.4.6 MCAGuard [on | off]

Switches on or off the machine area guard, no collisions will be given.
This is used during the rendering process, where the job file is checked for collisions with the machine area and tool change area.

4.2.4.7 HomeIsEstop [on | off]

This allows to control the homeIsEstop feature.
When on, a EStop is generated when one of the home sensors activate.

4.2.4.8 Exec <external Program> <"parameter"> <Timeout In Ms.>

This allows to execute an external program within the interpreter and wait until finished. The return code of the program is returned in #5399

Example:

Exec "notepad.exe" "hallo.txt" 60000
This executes notepad.exe and waits 1 minute for it to finish.
An error is generated if it does not finish within given time.
The maximum time is 10 minutes.

4.3 MACRO FILE AND AUTOMATIC TOOL CHANGE

Whenever a G-Code file is loaded, also the file "macro.cnc" is loaded. In this file you may put your frequently used subroutines, these can be invoked by the G-Code file through GOSUB subroutineName.
The file contains default one special subroutine called **"change_tool"**, this function is called automatically when a **M6 Tx** command (Tool change) is encountered in the G-Code file. With this it is possible to define your own tool change, especially useful when you have an automatic tool changer. You can put moves and I/O actions there as well as automatic tool length measurement using the probe with G38.2.

The tool change area can be guarded for collision, if it is defined the rendering process will detect eventual collisions and report it. So a normal workpiece program is not allowed to go through the Tool change Area.
The tool change itself is allowed to go to this area. Therefor the **change_tool** subroutine contains the statement **TCAGuard off** at the beginning and **TCAGuard on** at the end.

4.3.1 Tool change example

;* http://www.youtube.com/watch?v=_kp0SAeR-Kg

The grey marked code below is already prepared for you in the standard macro.cnc file.
This change_tool subroutine is automatically called when the interpreter encounters e.g.
m6 t1
(If automatic tool change in the setup is switched on).

```
;This example shows how to make your own tool_changer work.
;It is made for 6 tools and a simple KRESS Tool changer.
;First current tool is dropped, then the new tool is picked
;There is a check whether selected tool is already in the spindle
;Also a check that the tool is within 1-16
;There is a picktool subroutine for each tool and a droptool subroutine for each tool.
;These routines need to be modified to fit your machine and tool changer



sub change tool
    ;Switch off guard for tool change area collision
    TCAGuard off

   ;Switch off spindle
    m5

    ;Use #5015 to indicate succesfull toolchange
        #5015 = 0 ; Tool change not performed

    if [ [#5011] == [#5008] ]
       msg "Tool already in spindle"
    endif

    ; check tool in spindle and exit sub
    If [ [#5011] <> [#5008] ]
        if [[#5011] > 6 ]
            ; our tool changer supports 6 tools 1-6
            errmsg "Please select a tool from 1 to 6."
        else
            ;Drop current tool
            If [[#5008] == 0]
                GoSub DropTool0
            endif
            If [[#5008] == 1]
                GoSub DropTool1
            endif
            If [[#5008] == 2]
                GoSub DropTool2
            endif
            If [[#5008] == 3]
                GoSub DropTool3
            endif
            If [[#5008] == 4]
                GoSub DropTool4
            endif
            If [[#5008] == 5]
                GoSub DropTool5
            endif
```

```
            If [[#5008] == 6]
                GoSub DropTool6
            endif


            ;Pick new tool
            if [[#5011] == 0]
                GoSub PickTool0
            endif
            if [[#5011] == 1]
                GoSub PickTool1
            endif
            if [[#5011] == 2]
                GoSub PickTool2
            endif
            if [[#5011] == 3]
                GoSub PickTool3
            endif
            if [[#5011] == 4]
                GoSub PickTool4
            endif
            if [[#5011] == 5]
                GoSub PickTool5
            endif
            if [[#5011] == 6]
                GoSub PickTool6
            endif

        endif
    endif

    If [[#5015] == 1]
        msg "Tool "#5008" Replaced by tool "#5011" Tool length compensation G43 switched on"
        m6t[#5011]

        if [#5011 <> 0]
            G43 ;Use tool-length compensation
        endif
    else
        errmsg "tool change failed"
    endif

    ;Switch on guard for tool change area collision
    TCAGuard on


EndSub
```

The code below is also inside file macro.cnc, for each tool there is a DropTool sub-routine and a Pick-tool sun-routine. DropTool makes the movements and IO for putting the tool from spindle in the tool holder. PickTool makes the movements and I/O to pick the tool from the tool holder and put it in the spindle.

These subroutines below are machine dependent and need to be filled in by yourself.
This example works only for the Kress Tool changer.

```
;Drop tool subroutines, these put the tool in spindle in the tool-holder

Sub DropTool0
    msg "Dropping tool 0"
    ;Tool 0 is no tool, so we just open yhe tool station here for PickTool which comes next
    M54P3                       ;Open toolstation OUTPUT AUX3
    G4P1                        ;Wait 1 seconds
endsub

Sub DropTool1
    msg "Dropping tool 1"
    M5
    G53G0Z100                   ;Z up 110 is and machinebed is zero
    G53G0X825.00Y173.00         ;Move just before drop place
    M54P3                       ;Open toolstation
    G4P1                        ;Wait 1 seconds
    G53G1X867.206Y173.156F220   ;Move into drop place
    g53G0Z60                    ;Move down fast but not fully to the endposition
    g53G01Z32.000F120           ;Move down the last mm slower
    M54P1                       ;AUX1 ON air pressure toolchange
    G4P1                        ;Wait 1 second
    G53G01Z45F100               ;Move up slowly to move free from toolstation
    M55P1                       ;AUX1 off, tool dropped
    G53G0Z100                   ;Further up and done with dropping tool
endsub

Sub DropTool2
    msg "Dropping tool 2"
    M5
    G53G0Z100                   ;Z up 110 is and machinebed is zero
    G53G0X825.00Y208.00         ;Move just before drop place
    M54P3                       ;Open toolstation
    G4P01                       ;Wait 1 seconds
    G53G1X867.200Y208.156F220   ;Move into drop place
    g53G0Z60                    ;Move down fast but not fully to the endposition
    g53G01Z32.000F120           ;Move down the last mm slower
    M54P1                       ;AUX1 ON air pressure toolchange
    G4P1                        ;Wait 1 second
    G53G01Z45F100               ;Move up slowly to move free from toolstation
    M55P1                       ;AUX1 off, tool dropped
    G53G0Z100                   ;Further up and done with dropping tool
endsub

Sub DropTool3
    msg "Dropping tool 3"
    M5
    G53G0Z100                   ;Z up 110 is and machinebed is zero
    G53G0X825.00Y243.00         ;Move just before drop place
    M54P3                       ;Open toolstation
    G4P01                       ;Wait 1 seconds
    G53G1X867.116Y243.156F220   ;Move into drop place
    G53G0Z60                    ;Move down fast but not fully to the endposition
    G53G01Z32.000F120           ;Move down the last mm slower
    M54P1                       ;AUX1 ON air pressure toolchange
    G4P1                        ;Wait 1 second
    G53G01Z45F100               ;Move up slowly to move free from toolstation
    M55P1                       ;AUX1 off, tool dropped
    G53G0Z100                   ;Further up and done with dropping tool
endsub

Sub DropTool4
    msg "Dropping tool 4"
    M5
```

```
    G53G0Z100                   ;Z up 110 is and machinebed is zero
    G53G0X825.00Y278.00         ;Move just before drop place
    M54P3                       ;Open toolstation
    G4P01                       ;Wait 1 seconds
    G53G1X867.026Y278.156F220   ;Move into drop place
    g53G0Z60                    ;Move down fast but not fully to the endposition
    g53G01Z32.000F120           ;Move down the last mm slower
    M54P1                       ;AUX1 ON air pressure toolchange
    G4P1                        ;Wait 1 second
    G53G01Z45F100               ;Move up slowly to move free from toolstation
    M55P1                       ;AUX1 off, tool dropped
    G53G0Z100                   ;Further up and done with dropping tool
endsub

Sub DropTool5
    msg "Dropping tool 5"
    M5
    G53G0Z100                   ;Z up 110 is and machinebed is zero
    G53G0X825.00Y313.00         ;Move just before drop place
    M54P3                       ;Open toolstation
    G4P01                       ;Wait 1 seconds
    G53G1X866.936Y313.156F220   ;Move into drop place
    g53G0Z60                    ;Move down fast but not fully to the endposition
    g53G01Z32.000F120           ;Move down the last mm slower
    M54P1                       ;AUX1 ON air pressure toolchange
    G4P1                        ;Wait 1 second
    G53G01Z45f100               ;Move up slowly to move free from toolstation
    M55P1                       ;AUX1 off, tool dropped
    G53G0Z100                   ;Further up and done with dropping tool
endsub

Sub DropTool6
    msg "Dropping tool 6"
    M5
    G53G0Z100                   ;Z up 110 is and machinebed is zero
    G53G0X825.00Y348.00         ;Move just before drop place
    M54P3                       ;Open toolstation
    G4P01                       ;Wait 1 seconds
    G53G1X866.850Y348.156F220   ;Move into drop place
    g53G0Z60                    ;Move down fast but not fully to the endposition
    g53G01Z32.000F120           ;Move down the last mm slower
    M54P1                       ;AUX1 ON air pressure toolchange
    G4P1                        ;Wait 1 second
    G53G01Z45f100               ;Move up slowly to move free from toolstation
    M55P1                       ;AUX1 off, tool dropped
    G53G0Z100                   ;Further up and done with dropping tool
endsub
```

```
;Pick tool subroutines
Sub PickTool0
    msg "Picking tool 0"        ;Tool 0 is nothing, just close the tool station
    M55P3                       ;OUT3 off, closes toolstation
    #5015 = 1                   ;toolchange succes
endsub


Sub PickTool1
    msg "Picking tool 1"
    M5                          ;Be sure that spindle is off
    G53G0Z100                   ;Z up where zero is machinebed and 110 is top
    G53G0X825.00Y173.00         ;Move before pick place
    M54P3                       ;Open toolstation
    G4P01                       ;Wait 1 second
    G53G1X867.206Y173.156F220   ;Move into pick place
    G53G0Z50                    ;Move down fast but not fully to the end
    M54P1                       ;AUX1 ON for opening collet clamp
    G4P01                       ;wait 1 second
    G53G01Z23.000F120           ;Move down last mm down slower to pick up toolholder
    G4P0.5                      ;Wait 1 second
    M55P1                       ;AUX1 off, tool picked
    G53G01Z60F120               ;Move slowly up to pick up tool and move free
    G53G0X825Z100               ;Further up and done with dropping
    M55P3                       ;OUT3 off, closes toolstation
    #5015 = 1                   ;toolchange succes
endsub

Sub PickTool2
```

```
    msg "Picking tool 2"
    M5                          ;Be sure that spindle is off
    G53G0z100                   ;Z up where zero is machinebed and 110 is top
    G53G0X825.00Y208.00         ;Move before pick place
    M54P3                       ;Open toolstation
    G4P01                       ;Wait 1 second
    G53G1X867.200Y208.156F220   ;Move into drop place
    g53G0Z50                    ;Move down fast but not fully to the end
    M54P1                       ;AUX1 ON for opening collet clamp
    G4P1                        ;wait 1 second
    G53G01Z23.000F120           ;Move down last mm down slower to pick up toolholder
    G4P0.5                      ;Wait 1 second
    M55P1                       ;AUX1 off, tool picked
    G53G01Z60F120               ;Move slowly up to pick up tool and move free
    G53G0X825Z100               ;Further up and done with dropping
    M55P3                       ;OUT3 off, closes toolstation
    #5015 = 1                   ; toolchange succes
endsub

Sub PickTool3
    msg "Picking tool 3"
    M5                          ;Be sure that spindle is off
    G53G0z100                   ;Z up where zero is machinebed and 110 is top
    G53G0X825.00Y243.00         ;Move before pick place
    M54P3                       ;Open toolstation
    G4P01                       ;Wait 1 second
    G53G1X867.116Y243.156F220   ;Move into drop place
    G53G0Z50                    ;Move down fast but not fully to the end
    M54P1                       ;AUX1 ON for opening collet clamp
    G4P1                        ; wait 1 second
    G53G01Z23.000F120           ;Move down last mm down slower to pick up toolholder
    G4P01                       ;Wait 1 second
    M55P1                       ;AUX1 off, tool picked
    G53G01Z60F120               ;Move slowly up to pick up tool and move free
    G53G0X825Z100               ;Further up and done witg dropping
    M55P3                       ;OUT3 off, closes toolstation
    #5015 = 1                   ; toolchange succes
endsub

Sub PickTool4
    msg "Picking tool 4"
    M5                          ;Be sure that spindle is off
    G53G0z100                   ;Z up where zero is machinebed and 110 is top
    G53G0X825.00Y278.00         ;Move before pick place
    M54P3                       ;Open toolstation
    G4P01                       ;Wait 1 second
    G53G1X867.026Y278.156F220   ;Move into drop place
    g53G0Z50                    ;Move down fast but not fully to the end
    M54P1                       ;AUX1 ON for opening collet clamp
    G4P1                        ; wait 1 second
    G53G01Z23.000F120           ;Move down last mm down slower to pick up toolholder
    G4P0.5                      ;Wait 1 second
    M55P1                       ;AUX1 off, tool picked
    G53G01Z60F120               ;Move slowly up to pick up tool and move free
    G53G0X825Z100               ;Further up and done witg dropping
    M55P3                       ;OUT3 off, closes toolstation
    #5015 = 1                   ; toolchange succes
endsub

Sub PickTool5
    msg "Picking tool 5"
    M5                          ;Be sure that spindle is off
    G53G0z100                   ;Z up where zero is machinebed and 110 is top
    G53G0X825.00Y313.00         ;Move before pick place
    M54P3                       ;Open toolstation
    G4P01                       ;Wait 1 second
    G53G1X866.936Y313.156F220   ;Move into drop place
    G53G0Z60                    ;Move down fast but not fully to the end
    M54P1                       ;AUX1 ON for opening collet clamp
    G4P1                        ;wait 1 second
    G53G01Z23.000F120           ;Move down last mm down slower to pick up toolholder
    G4P0.5                      ;Wait 1 second
    M55P1                       ;AUX1 off, tool picked
    G53G01Z60F120               ;Move slowly up to pick up tool and move free
    G53G0X825Z100               ;Further up and done witg dropping
    M55P3                       ;OUT3 off, closes toolstation
    #5015 = 1                   ; toolchange succes
```

```
endsub

Sub PickTool6
    msg "Picking tool 6"
    M5
    M5                          ;Be sure that spindle is off
    G53G0z100                   ;Z up where zero is machinebed and 110 is top
    G53G0X825.00Y348.00         ;Move before pick place
    M54P3                       ;Open toolstation
    G4P01                       ;Wait 1 second
    G53G1X866.850Y348.156F220   ;Move into drop place
    g53G0Z60                    ;Move down fast but not fully to the end
    M54P1                       ;AUX1 ON for opening collet clamp
    G4P1                        ;wait 1 second
    g53G01Z23.000F120           ;Move down last mm down slower to pick up toolholder
    G4P0.5                      ;Wait 1 second
    M55P1                       ;AUX1 off, tool picked
    G53G01Z60F120               ;Move slowly up to pick up tool and move free
    G53G0X825Z100               ;Further up and done witg dropping
    M55P3                       ;OUT3 off, closes toolstation
    #5015 = 1                   ;toolchange succes
endsub
```

4.3.2 **USER Reset**

A subroutine called user_reset can be added to macro.cnc.
This allows to perform extra reset actions, e.g. set/reset IO using M54/M55.

Example of user reset, this one toggles AUX1 for resetting servo drives.

```
;Remove comments if you want additional reset actions
;when reset button was pressed in UI
sub user_reset
    m54 p1
    g4 p0.1
    m55 p1
    msg "Ready for operation"
endsub
```

4.4 RUN BEHAVIOR DURING SIMULATION AND RENDER

Simulation is the software mode when there is no hardware connected.
Render is performed during loading of a g-code file, the file is run by the interpreter while generating output for the graphic. During this no actions to the machine are applied, no motion and no IO, only checks if the g-code is valid and stay's within the machine limits.
If there is advanced macro programming, e.g. because the machine has automatic tool change, there it is important to take into account that in simulation and render mode no machine actions take place.

4.4.1 Example a check with error that we want to see always

Suppose we have a machine with a 6 tool automatic tool changer. A check is programmed hat generates an error if a g-code file is loaded that tries to change tool to a number that we do not have:

```
if [#5011 > 6]
  ErrMsg "Please select a tool in range 1-6"
Else
  ..
  ;Code to perform the tool change
Endif
```

It is logic that we want this "ErrMsg …" line to be executed always, when running but also in simulation mode and when loading the file.

4.4.2 Example a check with error showing only when running

```
sub check_airpresure
    m56 p5
    if [#5399 == 0]
      errmsg "Error, No Air pressure"
    else
      msg "Air Presure OK"
    endif
endsub
```

This check on air pressure will not work correctly while loading a g-code file or while the system is in simulation mode, because no actual inputs are read.

```
sub check_airpresure
    ;;Check only if not rendering and not simulation
    if [[#5380 == 0] and [#5397 == 0]]
        m56 p5
        if [#5399 == 0]
          errmsg "Error, No Air pressure"
        else
          msg "Air Presure OK"
        endif
    endif
endsub
```

Here we see the modification to make the on the air pressure sensor input 5 simulation and render proof, the check is only performed when we are not in render or simulation mode, only an extra if statement that checks #5380 (1 when simulation mode) and #5397 (1 when rendering) is added.

5 Cutter Radius Compensation

This appendix discusses cutter radius compensation. It is intended for NC programmers and machine operators.

See chapter 5 for additional information on cutter radius compensation.

5.1 INTRODUCTION

The cutter radius compensation capabilities of the Interpreter enable the programmer to specify that a cutter should travel to the right or left of an open or closed contour in the XY-plane composed of arcs of circles and straight line segments.

Cutter radius compensation is performed only with the XY-plane active. All the figures in this appendix, therefore, show projections on the XY-plane.

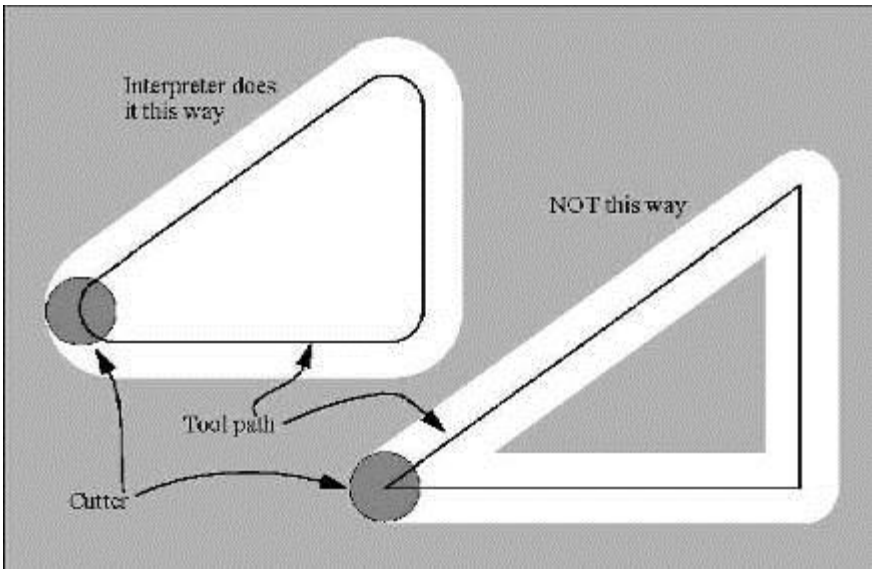
Where the adjacent sides of remaining material meet at a corner, there are two common ways to handle the tool path. The tool may pass in an arc around the corner, or the tool path may continue straight in the direction it was going along the first side until it reaches a point where it changes direction to go straight along the second side. Figure A-1 shows these two types of path. On Figure A-1:

- Uncut material is shaded in the figures. Note that the inner triangles have the same shape with both tool paths.
- The white areas are the areas cleared by the tool.
- The lines in the center of the white areas represent the path of the tip of a cutting tool.
- The tool is the cross-hatched circles.

Both paths will clear away material near the shaded triangle and leave the shaded triangle uncut. When the Interpreter performs cutter radius compensation, the tool path is rounded at the corners, as shown on the left in Figure A-1. In the method on the right (the one not used), the tool does not stay in contact with the shaded triangle at sharp corners, and more material than necessary is removed.

There are also two alternatives for the path that is programmed in NC code during cutter radius compensation. The programmed path may be either (1) the edge of the material to remain uncut (for example, the edge of the inner triangle on the left of Figure A-1), or (2) the nominal tool path (for example, the tool path on the left side of Figure A-1). The nominal tool path is the path that would be used if the tool were exactly the intended size. The Interpreter will handle both cases without being told which one it is. The two cases are very similar, but different enough that they are described in separate sections see below.

Figure A-1



Z-axis motion may take place while the contour is being followed in the XY-plane. Portions of the contour may be skipped by retracting the Z-axis above the part, following the contour to the next point at which machining should be done, and re-extending the Z-axis. These skip motions may be performed at feed rate (G1) or at traverse rate (G0). The Z motion will not interfere with the XY path following. The sample NC code in this appendix does not include moving the Z-axis. In actual programs, include Z-axis motion wherever you want it.

Rotational axis motions (A, B, and C axes) are allowed with cutter radius compensation, but using them would be very unusual.

Inverse time feed rate (G93) or units per minute feed rate (G94) may be used with cutter radius compensation. Under G94, the feed rate will apply to the actual path of the cutter tip, not to the programmed contour.

5.1.1 Data for Cutter Radius Compensation

The Interpreter world model keeps three data items for cutter radius compensation: the setting itself (right, left, or off), program_x, and program_y. The last two represent the X and Y positions which are given in the NC code while compensation is on. When compensation is off, these both are set to a very small number (10-20) whose symbolic value is "unknown". The Interpreter world model uses the data items current_x and
current_y to represent the position of the center of the tool tip (in the currently active coordinate system) at all times.

5.2 PROGRAMMING INSTRUCTIONS

5.2.1 Turning Cutter Radius Compensation On

To start cutter radius compensation keeping the tool to the left of the contour, program **G41 D...**. The D word is optional (see "Use of D Number", just below).

To start cutter radius compensation keeping the tool to the right of the contour, program **G42 D...** .

In Figure A-1, for example, if G41 were programmed, the tool would move clockwise around the triangle, so that the tool is always to the left of the triangle when facing in the direction of travel. If G42 were programmed, the tool would stay right of the triangle and move counter clockwise around the triangle.

5.2.2 Turning Cutter Radius Compensation Off

To stop cutter radius compensation, program **G40**. It is OK to turn compensation off when it is already off.

5.2.3 Sequencing

If G40, G41, or G42 is programmed on the same line as tool motion, cutter
compensation will be turned on or off before the motion is made. To make the motion come first, the motion must be programmed on a separate, previous line of code.

5.2.4 Use of D Number

Programming a D word with G41 or G42, is optional.
If a D number is programmed, it must be a non-negative integer. It represents the slot number of the tool whose radius (half the diameter given in the tool table) will be used, or it may be zero (which is not a slot number). If it is zero, the value of the radius will also be zero. Any slot in the tool table may be selected. The D number does not have to be the same as the slot number of the tool in the spindle, although it is rarely useful for it not to be.
If a D number is not programmed, the slot number of the tool in the spindle will be used as the D number.

5.2.5 Material Edge Contour

When the contour is the edge of the material, the outline of the edge is described in the NC program.

For a material edge contour, the value for the diameter in the tool table is the actual value of the diameter of the tool. The value in the table must be positive. The NC code for a material edge contour is the same regardless of the (actual or intended) diameter of the tool.

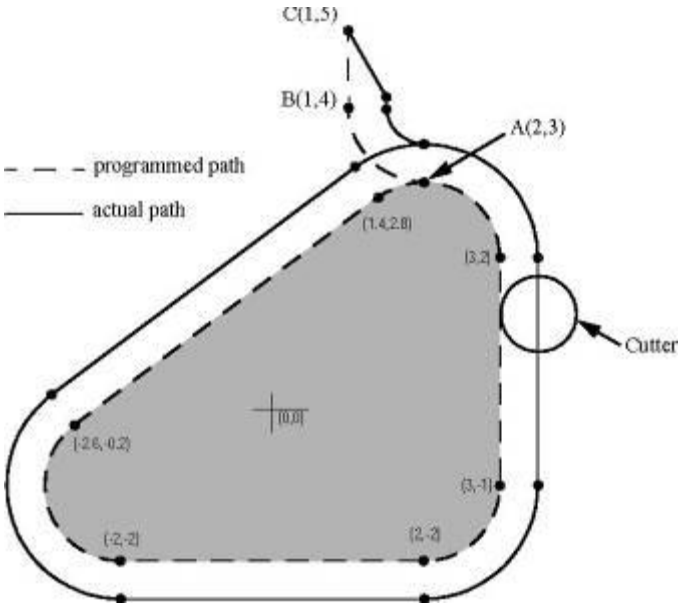
5.2.6 Programming Entry Moves

In general, two pre-entry moves and one entry move are needed to begin compensation correctly. However, if there is a convex corner on the contour, a simpler method is available using zero or one pre-entry move and one entry move. The general method,
which will work in all situations, is described first. We assume here that the programmer knows what the contour is already and has the job of adding entry moves.

A.1.1.1 GENERAL METHOD

The general method includes programming two pre-entry moves and one entry move. See Figure A-2. The shaded area is the remaining material. It has no corners, so the simple method cannot be used. The dotted line is the programmed path. The solid line is the actual path of the tool tip. Both paths go clockwise around the remaining material. A cutter one unit in diameter is shown part way around the path. The black dots mark points at the beginning or end of programmed or actual moves. The figure shows the second pre-entry move but not the first, since the beginning point of the first pre-entry move could be anywhere.

Figure A-2, Cutting radius compensation entry moves (for material edge contour)



First, pick a point A on the contour where it is convenient to attach an entry arc. Specify an arc outside the contour which begins at a point B and ends at A tangent to the contour (and going in the same direction as it is planned to go around the contour). The radius of the arc should be larger than half the diameter given in the tool table. Then extend a line tangent to the arc from B to some point C, located so that the line BC is more than one tool radius long. After the construction is finished, the code is written in the reverse order from the construction. The NC code is shown in Table A-1; the first three lines are the entry moves just described.

Table A-1 NC program for figure A-2

N0010 G1 X1 Y5 (make first pre-entry move to C)
N0020 G41 G1 Y4 (turn compensation on and make second pre-entry move to point B)
N0030 G3 X2 Y3 I1 (make entry move to point A)
N0040 G2 X3 Y2 J-1 (cut along arc at top)
N0050 G1 Y-1 (cut along right side)
N0060 G2 X2 Y-2 I-1 (cut along arc at bottom right)
N0070 G1 X-2 (cut along bottom side)
N0080 G2 X-2.6 Y-0.2 J1 (cut along arc at bottom left)
N0090 G1 X1.4 Y2.8 (cut along third side)
N0100 G2 X2 Y3 I0.6 J-0.8 (cut along arc at top of tool path)
N0110 G40 (turn compensation off)

Cutter radius compensation is turned on after the first pre-entry move and before the second pre-entry move (including G41 on the same line as the second pre-entry move turns compensation on before the move is made). In the code above, line N0010 is the first pre-entry move, line N0020

turns compensation on and makes the second pre-entry move, and line N0030 makes the entry move.

A.1.1.2 SIMPLE METHOD

If there is a convex (sticking out, not in) corner somewhere on the contour, a simpler method of making an entry is available. See Figure A-3.

First, pick a convex corner. There is only one corner in Figure A-3. It is at A, and it is convex. Decide which way you want to go along the contour from A. In our example we are keeping the tool to the left of the remaining material and going clockwise. Extend the side to be cut (DA in the figure) to divide the area outside the material near A into two regions; DA extended is the dotted line AC on the figure. Make a pre-entry move to anywhere in the region on the same side of DC as the remaining material (point B on the figure) and not so close to the remaining material that the tool is cutting into it. Anywhere in the diagonally shaded area of the figure (or above or to the left of that area) is OK. If the tool is already in region, no pre-entry move is needed. Write a line of NC code to move to B, if necessary. Then write a line of NC code for a straight entry move that turns compensation on and goes to point A.
If B is at (1.5, 4), the two lines of code for the pre-entry and entry moves would be:

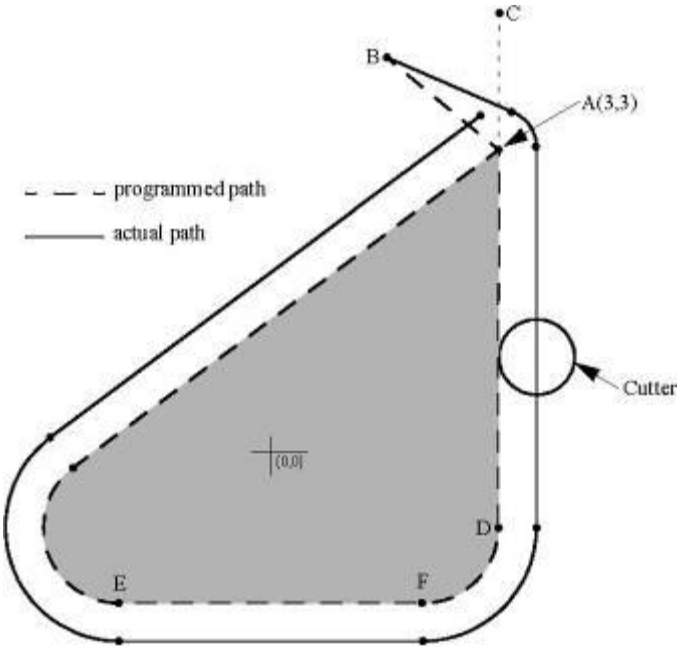
N0010 G1 X1.5 Y4 (move to B)
N0020 G41 G1 X3 Y3 (turn compensation on and make entry move to A)

These two lines would be followed by four lines identical to lines N0050 to N0080 from Table A-1, but the end of the program would be different since the shape of remaining material is different.

It would be OK for B to be on line AC. In fact, B could be placed on the extension outside the part of any straight side of the part. B could be placed on EF extended to the right (but not to the left, for going clockwise), for example.

If DA were an arc, not a straight line, the two lines of code above would still be suitable. In this case, the dotted line extending DA should be tangent to DA at A.
Figure A-3 Simpler cutter radius compensation entry move (for material edge contour)

Figure A-3 Simpler compensation entry move



5.3 NOMINAL PATH CONTOUR

When the contour is a nominal path contour (the path a tool with exactly the intended diameter would take), the tool path is described in the NC program. It is expected that (except for during the entry moves) the path is intended to create some part geometry. The path may be generated manually or by a post-processor, considering the part geometry which is intended to be made. For the Interpreter to work, the tool path must be such that the tool stays in contact with the edge of the part geometry, as shown on the left side of Figure A-1. If a path of the sort shown on the right of Figure A-1 is used, in which the tool does not stay in contact with the part geometry all the time, the Interpreter will not be able to compensate properly when undersized tools are used. A nominal path contour has no corners, so the simple method just described will not work.

For a nominal path contour, the value for the cutter diameter in the tool table will be a small positive number if the selected tool is slightly oversized and will be a small negative number if the tool is slightly undersized. If a cutter diameter value is negative, the Interpreter compensates on the other side of the contour from the one programmed and uses the absolute value of the given diameter. If the actual tool is the correct size, the value in the table should be zero. Suppose, for example, the diameter of the cutter currently in the spindle is 0.97, and the diameter assumed in generating the tool path was 1.0. Then the value in the tool table for the diameter for this tool should be -0.03.

The nominal tool path needs to be programmed so that it will work with the largest and smallest tools expected to be actually used. We will call the difference between the radius of the largest expected tool and the intended radius of the tool the "maximum
radius difference." This is usually a small number.

The method includes programming two pre-entry moves and one entry moves. See Figure A-4. The shaded area is the remaining material. The dashed line is the programmed tool path. The solid line is the actual path of the tool tip. Both paths go clockwise around the remaining material. The actual path is to the right of the programmed path even though G41 was programmed, because the diameter value is negative. On the figure, the distance between the two paths is larger than would normally be expected. The 1-inch diameter tool is shown part way around the path. The black dots mark points at the beginning or end of programmed moves. The corresponding points on the actual path have not been marked. The actual path will have a very small additional arc near point B unless the tool diameter is exactly the size intended. The figure shows the second pre-entry move but not the first, since the beginning point of the first pre-entry move could be anywhere.

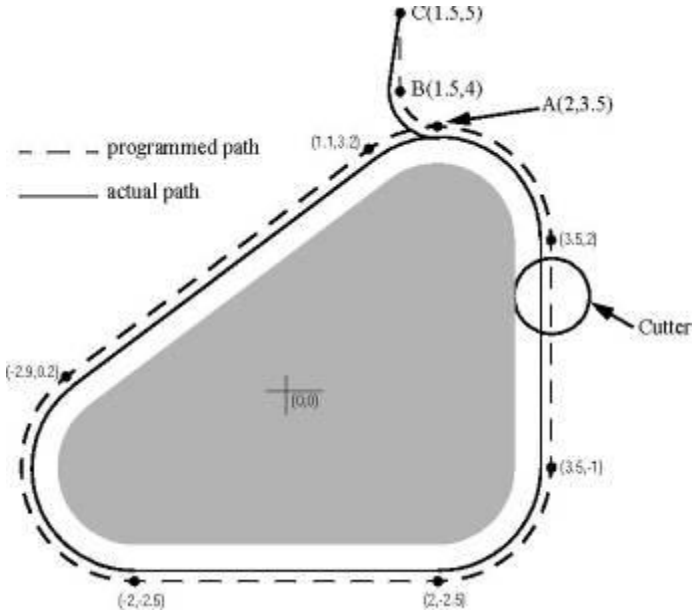
First, pick a point A on the contour where it is convenient to attach an entry arc. Specify an arc outside the contour which begins at a point B and ends at A tangent to the contour (and going in the same direction as it is planned to go around the contour). The radius of the arc should be larger than the maximum radius difference. Then extend a line tangent to

the arc from B to some point C, located so that the length of line BC is more than the maximum radius difference. After the construction is finished, the code is written in the reverse order from the construction. The NC code is shown in Table A-2; the first three lines are the entry moves just described.

Table A-2 NC program for Figure A-4

N0010 G1 X1.5 Y5 (make first pre-entry move to C)
N0020 G41 G1 Y4 (turn compensation on and make second pre-entry move to point B)
N0030 G3 X2 Y3.5 I0.5 (make entry move to point A)
N0040 G2 X3.5 Y2 J-1.5 (cut along arc at top)
N0050 G1 Y-1 (cut along right side)
N0060 G2 X2 Y-2.5 I-1.5 (cut along arc at bottom right)
N0070 G1 X-2 (cut along bottom side)
N0080 G2 X-2.9 Y0.2 J1.5 (cut along arc at bottom left)
N0090 G1 X1.1 Y3.2 (cut along third side)
N0100 G2 X2 Y3.5 I0.9 J-1.2 (cut along arc at top of tool path)
N0110 G40 (turn compensation off)

Figure A-4 Cutter radius compensation entry moves



Cutter radius compensation is turned on after the first pre-entry move and before the second pre-entry move (including G41 on the same line as the second pre-entry move turns compensation on before the move is made). In the code above, line N0010 is the first pre-entry move, line N0020 turns compensation on and makes the second pre-entry move, and line N0030 makes the entry move.

5.4 PROGRAMMING ERRORS AND LIMITATIONS

The Interpreter will issue the following error messages involving cutter radius compensation. In addition to these, there are several bug messages related to cutter compensation, but they should never occur.

1. Cannot change axis offsets with cutter radius comp
2. Cannot change units with cutter radius comp
3. Cannot probe with cutter radius comp on
4. Cannot turn cutter radius comp on out of xy-plane
5. Cannot turn cutter radius comp on when on
6. Cannot use g28 or g30 with cutter radius comp
7. Cannot use g53 with cutter radius comp
8. Cannot use xz-plane with cutter radius comp
9. Cannot use yz-plane with cutter radius comp
10. Concave corner with cutter radius comp
11. Cutter gouging with cutter radius comp
12. D word with no g41 or g42
13. Multiple d words on one line
14. Negative d word tool radius index used
15. Tool radius index too big
16. Tool radius not less than arc radius with comp
17. Two g codes used from same modal group.

Most of these are self-explanatory. For those that require explanation, an explanation is given below.

Changing a tool while cutter radius compensation is on is not treated as an error, although it is unlikely this would be done intentionally. The radius used when cutter radius compensation was first turned on will continue to be used until compensation is turned off, even though a new tool is actually being used.

A.1.2 Concave Corner and Tool Radius Too Big (10 and 16)

When cutter radius compensation is on, it must be physically possible for a circle whose radius is the half the diameter given in the tool table to be tangent to the contour at all points of the contour. In particular, the Interpreter treats concave corners and concave arcs into which the circle will not fit as errors, since the circle cannot be kept tangent to the contour in these situations. See Figure A-5. This error detection does not limit the shapes which can be cut, but it does require that the programmer specify the actual shape to be cut (or path to be followed), not an approximation. In this respect, the NIST
RS274/NGC Interpreter differs from interpreters used with many other controllers, which often allow these errors silently and either gouge the part or round the corner.

concave corner
-tool does not fit

concave arc too small
-tool does not fit

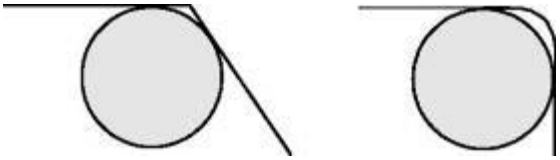


Figure A-5, Two cutter radius compensation errors

In both examples, the line represents a contour, and the circle represents the cross section of a tool following the contour using cutter radius compensation (tangent to one side of the path.)

A.1.3 Cannot Turn Cutter Radius Comp on When On (5)

If cutter radius compensation has already been turned on, it cannot be turned on again. It must be turned off first; then it can be turned on again. It is not necessary to move the
cutter between turning compensation off and back on, but the move after turning it back on will be treated as a first move, as described below.
It is not possible to change from one cutter radius index to another while compensation is on because of the combined effect of rules 5 and 12. It is also not possible to switch compensation from one side to another while compensation is on.

A.1.4 Cutter Gouging (11)

If the tool is already covering up the next XY destination point when cutter radius compensation is turned on, the gouging message is given when the line of NC code
which gives the point is reached. In this situation, the tool is already cutting into material it should not cut. More details are given in Section A.6.

A.1.5 Tool Radius Index Too Big (15)

If a D word is programmed that is larger than the number of tool carousel slots, this error message is given. In the SAI, the number of slots is 68.

A.1.6 Two G Codes Used from Same Modal Group (17)

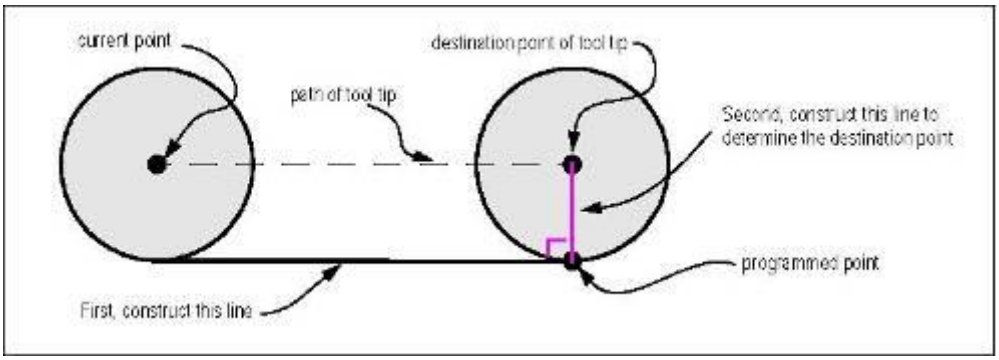
This is a generic message used for many sets of G codes. As applied to cutter radius
compensation, it means that more than one of G40, G41, and G42 appears on a line of NC code. This is not allowed.

A.2 First Move into Cutter Compensation

The algorithm used for the first move after cutter radius compensation is turned on, when the first move is a straight line, is to draw a straight line from the programmed destination point which is tangent to a circle whose center is at the current point and whose radius is the radius of the tool. The destination point of the tool tip is then found as the center of a circle of the same radius tangent to the tangent line at the destination point. If the programmed point is inside the initial cross section of the tool (the circle on the left), an error is signaled. The concept of the algorithm is shown in Figure A-6.
The function that locates the destination point actually takes a computational shortcut based on the fact that the line (not drawn on the

figure) from the current point to the programmed point is the hypotenuse of a right triangle having the destination point at the corner with the right angle.

Figure A-6, First cutter radius compensation move - Straight


current point
destination point of tool tip
path of tool tip
Second, construct this line to determine the destination point
programmed point
First, construct this line

If the first move after cutter radius compensation has been turned on is an arc, the arc which is generated is derived from an auxiliary arc which has its center at the programmed center point, passes through the programmed end point, and is tangent to the cutter at its current location. If the auxiliary arc cannot be constructed, an error is signaled. The generated arc moves the tool so that it stays tangent to the auxiliary arc throughout the move. This is shown in Figure A-7.

Figure A-7, First cutter radius compensation move - Arc

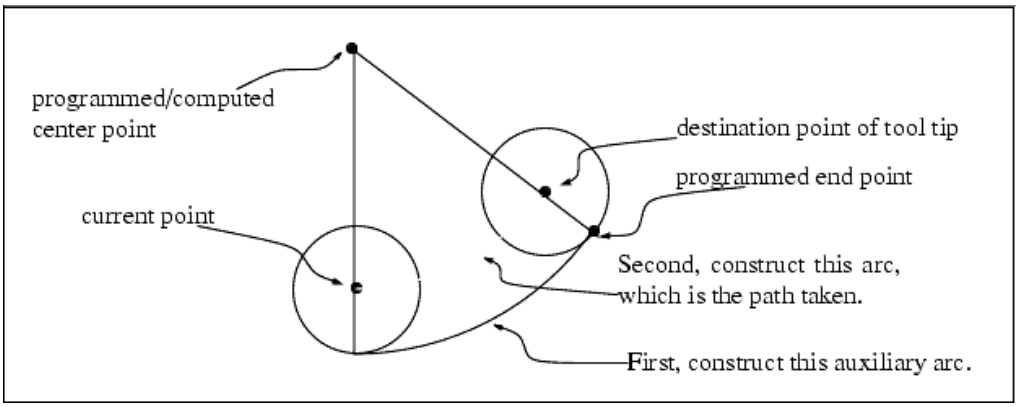
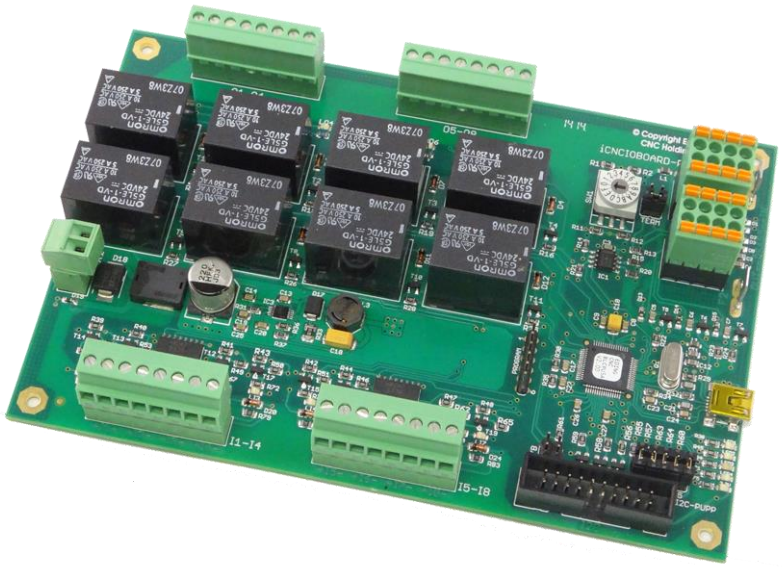

programmed/computed center point
destination point of tool tip
programmed end point
current point
Second, construct this arc, which is the path taken.
First, construct this auxiliary arc.

Figure A-7 shows the conceptual approach for finding the arc. The actual computations differ between the center format arc and the radius format arc (see Section 3.6.3).
After the entry moves of cutter radius compensation, the Interpreter keeps the tool tangent to the programmed path on the appropriate side. If a convex corner is on the path, an arc is inserted to go around the corner. The radius of the arc is half the diameter given in the tool table.
When cutter radius compensation is turned off, no special exit move takes place. The next move is what it would have been if cutter radius compensation had never been turned on and the previous move had placed the tool at its current position.

5.5 EXTENSION IO BOARD



The extension I/O board is not so often used, so all related settings are to be set directly in the cnc.ini file.

Special for GPIO card settings:

If you have an additional General purpose IO board, such as the RLY8 board with 8 output relays and 8 opto-isolated inputs, there are extra options currently only available by editing the cnc.ini file. This is the file soring all the parameters of the machine.

The CPIO card can guard the inputs and an action can be coupled when a guard triggers. There is a text that is displayed by the GUI when a guard triggers. You specify which inputs must be guards, this is given by a bit mask.
1 means only input 1.
3 means input 1 and input 2.
This table apply' s for input 0-3, the total table would be 255 lines long:

Participant bits Input 8 .. Input 1	Value	Participant bits Input 8 .. Input 1	Value
0000 0000 (-)	0	0100 0000 (I7)	64
0000 0001 (I1)	1	1000 0000 (I8)	128
0000 0010 (I2)	2		
0000 0100 (I3)	4	Example (I4 & I5)	
0000 1000 (I4)	8	0000 1000	
0001 0000 (I5)	16	0001 0000	Add =
0010 0000 (I6)	32	0001 1000	24

Next you specify the compare value if the guard should trigger when an input becomes ON (1), specify a 1 for that input, for OFF specify a zero.

Next, the action can be specified:

0 = nothing.
1 = warning.
2 = smooth stop (with ramp down).
3 = quick stop (immediately, no ramp down, this may give position loss.

This is how it looks in the cnc.ini for 1 rule,
there are max 8 rules for the RLY8 card.

```
[GPIO_RULES]
;Give warning message when Oil pressure 1 or 2 is low
;Input 4 = 0000 1000, input 5 = 0001 0000 together 0001 1000
;So the value for participant bits is 8 + 16 = 24
;When one of the inputs becomes 1, we want a warning message
;So compare bits are also 24
;Action is 1, warning
card_1_rule_1_text = "---Warning Oil is pressure low---"
card_1_rule_1_inputParticipantBits = 24
card_1_rule_1_inputCompareBits = 24
card_1_rule_1_action = 1

;Give smooth stop when Vacuum is to low
;Input 1 = 0000 0001
;So the value for participant bits is 1
;When one of the inputs becomes 1, we want a smooth stop
;So compare bits are 1
;Action is 2, smooth stop
card_1_rule_2_text = "---Warning Vacuum is pressure low---"
card_1_rule_2_inputParticipantBits = 1
card_1_rule_2_inputCompareBits = 1
card_1_rule_2_action = 2

;Give ESTOP when the water pressure is low
;Input 2 = 0000 0010
;So the value for participant bits is 2
;When one of the inputs becomes 1, we want a smooth stop
;So compare bits are 1
;Action is 3, ESTOP
card_1_rule_3_text = "---Water pressure low---"
card_1_rule_3_inputParticipantBits = 2
card_1_rule_3_inputCompareBits = 2
card_1_rule_3_action = 3
```

What also can be specified is the name of the IO in or output for the GUI. It is also done directly in the cnc.ini.

```
[GPIO_NAMES]
gp_output_101_name = "Oil pump on"
gp_output_102_name = "Water pump on"
gp_output_103_name = "Touch Probe down"
gp_output_104_name = "Touch Probe up"
gp_output_105_name = "Vacuum 1"
gp_output_106_name = "Vacuum 2"
gp_output_107_name = "Vacuum 3"
gp_output_108_name = "Vacuum 4"
gp_input_101_name = "Vacuum pressure low"
gp_input_102_name = "No water pressure"
gp_input_103_name = ""
gp_input_104_name = "Oil pressure 1 low"
```

gp_input_105_name = "Oil pressure 2 low"
gp_input_106_name = ""
gp_input_107_name = ""
gp_input_108_name = ""

These are the IO's for card 1, the IO's for card 2 have numbers 201 etc. instead of 101, card 3 has numbers 301.. etc.
If a name is not filled in, the GUI will show the number.

5.6 HARDWARE INSTALLATION TIPS

Building a reliable CNC system is not just making the right connections. EMC plays an important role here. We have several components that generate a lot of EMC noise, the drivers, the power supplies if they are switched mode supplies, the VFD if we have a HF spindle. The CPU and the communication to the PC, especially USB is very sensitive for EMC and may stop functioning when we make "spaghetti" wiring and no good functional Earth. So the routing of the cabling must be done in a right way. Very important is making a good EMC functional Earth using a star point GND.

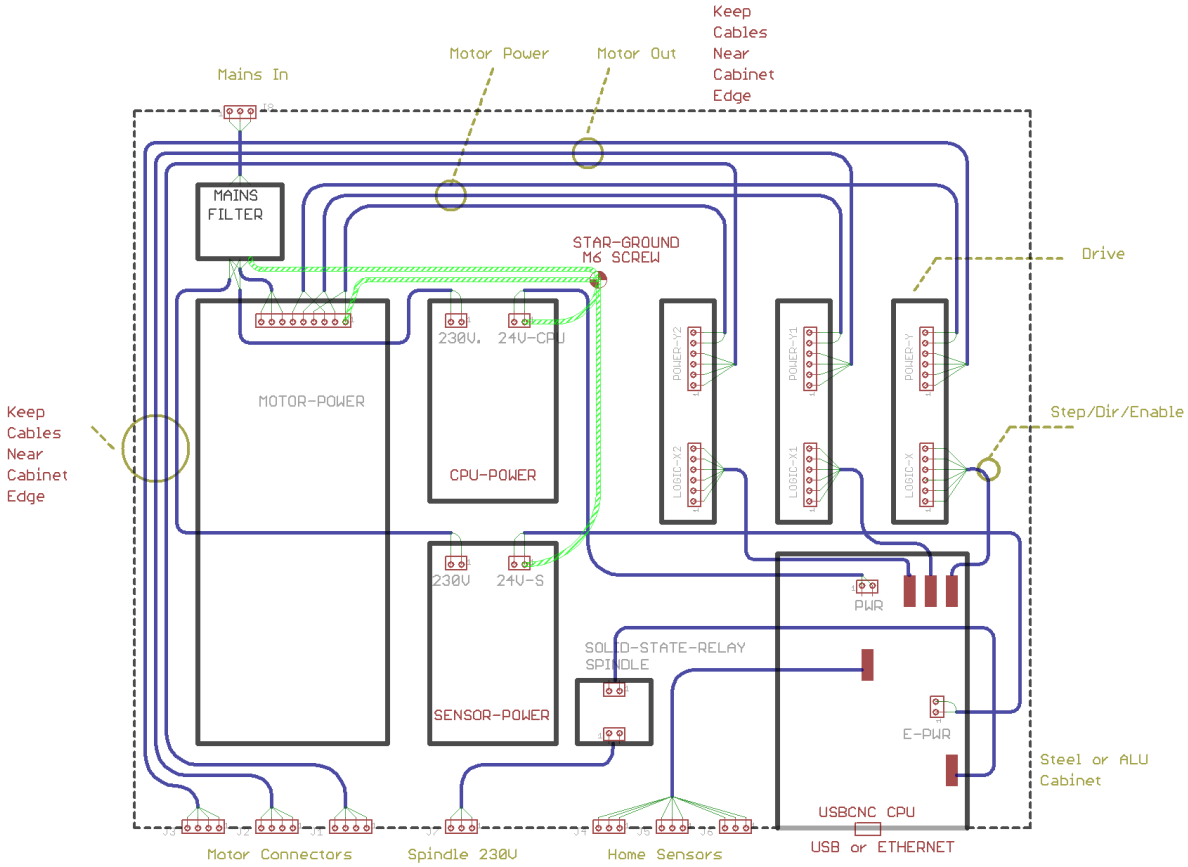
To prevent this miss function due to EMC, the EDINGCNC CPU should be build in correctly according these general EMC rules:

- Mount all electronics in a metal cabinet or on a metal plate in a plastic cabinet.
- Use a mains filter.
- Create a central GND point near the filter and connect the PE (Protective Earth) as well as the GND from all power supplies to this point.

- Route motor cables nicely along the cabinet edge, as far as possible away from the CPU. This way the cables noise radiation can flow away to the cabinet.
- Use shielded cables for the motor connections, both inside the cabinet and outside the cabinet. Connect the shield at one side to the central ground point, leave the other side un-connected.
- Use a professional USB2 cable, double shielded with ferrites like this:

- Keep all GND cables especially short and use thick flexible cable
- If not possible to keep it short, then connect it to the metal GND plate.

Schematic drawing of a possible good layout in the cabinet.



Here a picture of my own system, it contains various EMC problem makers, like 2 Switched mode power supplies and a frequency inverter for a HF spindle. Check the routing of the Motor and drive supply wires.

Also there a 4 stepper-motor drives working at 75 Volt, motor currents 4,2 Amp.

5.7 REFERENCES

[Albus] Albus, James S; et al; NIST Support to the Next Generation Controller Program: 1991 Final Technical Report; NISTIR 4888; National Institute of Standards and Technology, Gaithersburg, MD; July 1992

[Allen-Bradley] Allen-Bradley; RS274/NGC for the Low End Controller; First Draft; Allen-Bradley; August 1992

[EIA] Electronic Industries Association; EIA Standard EIA-274-D Interchangeable Variable Block Data Format for Positioning, Contouring, and Contouring/Positioning Numerically Controlled Machines; Electronic Industries Association; Washington, DC; February 1979

[Fanuc] Fanuc Ltd.; Fanuc System 9-Model A Operators Manual; Pub B-52364E/03; Fanuc Ltd; 1981

[Kramer1] Kramer, Thomas R.; Proctor, Frederick M.; Michaloski, John L.; The NIST RS274/NGC Interpreter, Version 1; NISTIR 5416; National Institute of Standards and Technology, Gaithersburg, MD; April 1994

[Kramer2] Kramer, Thomas R.; Proctor, Frederick M.; The NIST RS274KT Interpreter; NISTIR 5738; National Institute of Standards and Technology, Gaithersburg, MD; October 1995

[Kramer3] Kramer, Thomas R.; Proctor, Frederick M.; The NIST RS274/NGC Interpreter - Version 2; NISTIR 5739; National Institute of Standards and Technology, Gaithersburg, MD; October 1995

[Kramer4] Kramer, Thomas R.; Proctor, Frederick M.; The NIST RS274/VGER Interpreter; NISTIR 5754; National Institute of Standards and Technology, Gaithersburg, MD; November 1995

[K&T] Kearney and Trecker Co.; Part Programming and Operating Manual, KT/CNC Control, Type C; Pub 687D; Kearney and Trecker Corp.; 1980

[NCMS] National Center for Manufacturing Sciences; The Next Generation Controller Part Programming Functional Specification (RS-274/NGC); Draft; NCMS; August 1994

[Proctor] Proctor, Frederick M.; Kramer, Thomas R.; Michaloski, John L.; Canonical Machining Commands; NISTIR 5970; National Institute of Standards and Technology, Gaithersburg, MD; January 1997